A STUDY of GAS FLOW BEHAVIOR in AIRBAG DEPLOYMENT SIMULATION

Hitoshi Ida
Masashi Aoki
Michihisa Asaoka
Toyota Gosei Co., Ltd.
Japan

Kiyonobu Ohtani
Institute of Fluid Science, Tohoku University
Japan

Paper Number 15-0081

ABSTRACT

Airbag deployment simulation has been utilized as an important technique to predict the occupant protection performance in the development and design stages. One of the key elements of airbag deployment behavior is the gas flow behavior of jets from inflator. In this study, in order to understand the gas flow behavior of disk type inflator for driver side airbag, visualization experiments were conducted using the schlieren method. The gas flow from the inflator with a retainer has been found to have a strong directivity. Then, the gas flow simulation was conducted with a general purpose finite element program, LS-DYNA, it was possible to obtain a good reproducibility. For reproduction, it was found that jet direction and cone angle of gas diffusion were essential elements. Furthermore, comparison between simulation and experiments were conducted for deployment behavior of driver side airbag, the effect of gas flow on deployment behavior was analyzed. It was found from the results that the reproduction of gas flow from inflator was a major factor for reproduction on deployment behavior of driver side airbag.

INTRODUCTION

To evaluate the occupant protection performance of the airbag, the airbag deployment simulation is an important and efficient one approach. The first developed approach was uniform pressure method, which obtained pressure from mixed jet gas property of inflator and equation of state was applied to entire inside of airbag. This method could evaluate energy absorption of airbag and used for occupants protection analysis combined with kinematic analysis.[1] However, Since the gas flow was not considered in this method, there were some issues that behavior and energy absorption of airbag in deploying process could not be obtained accurately. To resolve the issues, fluid and structure coupling method, ALE (Arbitrary Lagrangian-Eulerian) method has been introduced.[2] When this method was applied to airbag deployment analysis, to represent the deployment of the folded airbag, enormous computational resources and cost were necessary.[3] To overcome the issue, a general purpose finite element program, LS-DYNA, implemented a new method CPM (Corpuscular Particle Method) which replace gas flow as particle movements. In this method, the gas was not treated as continuum and followed gas molecular dynamics. Instead of all models of the gas molecules, the overall translational kinetic energy are replaced with a number of particles to be equivalent.[4, 5] It was not necessary to discrete entire space as same as ALE in this method, the deployment simulation has been executed in available computational resources and cost. When the gas flow in narrow tube such as curtain airbag, it was possible to predict the deployment behavior and an impact force property. The prediction is currently applied to products development.[6] However, the difference has occurred in actual phenomena and simulation results when the gas was evolved in large space such as driver side airbag. We focused the gas flow in the airbag, tried to visualize the gas flow from inflator using schlieren method. In the past study, there was a observation of the gas flow only inside the inflator.[7] Very few attempts have been made at such observation of the gas flow outside the inflator for airbag deployment behavior.
In this paper, the visualization experiments of the jet gas flow were conducted and reproduced the gas flow by CPM. Then we applied the study results to deployment simulation of driver side airbag, and present the deployment behavior was reproduced properly.

**METHODS**

**Visualization Experiments of Gas Flow**

To understand the jet gas flow from inflator, the visualization experiment using schlieren method was conducted in open atmosphere space. It was hard to perceive clearly the jet gas flow with high speed camera. There was a PIV method to observe velocity of marker particles which mixed in gas flow.[8] In this method, observation area was local and it was hard to visualize the range of gas flow. Therefore, we selected the schlieren method to visualize clearly and directly. The schlieren method is one method of observation for gas flow using difference of light refractive index. The method has been used to visualize a shock wave of explosion or aircraft.[9,10] The configuration of experiment apparatus are shown in Figure 1. A light from point light source was parallelized by parabolic mirror, object inflator gas was ejected in the parallelized light. The light was condensed by parabolic mirror again. The defocused light by difference of refractive index was removed by iris at focal point. Images of difference of light contrasting were recorded with a high speed camera. In generaly, Although a knife edge was used in schlieren method, to observe the diffusing gas from the center of inflator, iris was used to capture the gas clearly.

To take a picture of gas flow in range of airbag deployment, the world’s largest class parabolic mirror with a diameter of one meter and focal length of eight meter was used in this experiments. This apparatus could be visualized at least 3kPa pressure waves such as sound from trombone.[11] For driver side airbag, the gas flow from inflator with or without a retainer was recorded with a high speed camera.

![Gas flow visualization apparatus of schlieren method at Tohoku University.](image)

**Figure 1.** Gas flow visualization apparatus of schlieren method at Tohoku University.
Gas Flow Simulation of Inflator

Gas flow simulation with or without a retainer using CPM of the LS-DYNA were conducted to reproduce the observed jet gas flow by visualization experiment. The research of seven CPM parameters is examined to reproduce the real gas flow. The examined parameters are shown in Table 1.

**Table 1. Parameters of gas flow simulation**

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Without retainer</th>
<th>With retainer</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Initial direction of gas inflow</td>
<td>Radial</td>
<td>Radial / Axial</td>
</tr>
<tr>
<td>2</td>
<td>Cone angle from orifices</td>
<td>Inactive / 16°/ 25°</td>
<td>Inactive / 0.1~25°</td>
</tr>
<tr>
<td>3</td>
<td>Friction factor</td>
<td>0 (default)</td>
<td>0 ~ -0.2</td>
</tr>
<tr>
<td>4</td>
<td>Dynamic scaling of particle</td>
<td>Inactive / Active</td>
<td>Inactive / Active</td>
</tr>
<tr>
<td>5</td>
<td>Initial gas inside airbag</td>
<td>CV method / Particle</td>
<td>CV method / Particle</td>
</tr>
<tr>
<td>6</td>
<td>Number of orifices</td>
<td>16 (Inflator pinholes)</td>
<td>16 / 4 (Retainer corner)</td>
</tr>
<tr>
<td>7</td>
<td>Number of gas components</td>
<td>Mixed / Multiple</td>
<td>Mixed / Multiple</td>
</tr>
</tbody>
</table>

Deployment Experiment of Driver Side Airbag

The static deployment behavior of driver side airbag was observed in an experiment. The experiment setting, the airbag configuration and deployment appearance are shown in Figure 2.
Figure 2. Static deployment experiment of driver side airbag.

The unfolded tether less airbag with retainer was installed for the experiment. A load cell was set under the retainer to measure a deployment force at installation point.

Deployment Simulation of Driver Side Airbag

A reproduce simulation shown in Figure 2 was conducted. A mechanical property of airbag fabric was from tensile, shear test and reflected to the property of input deck. A gas temperature and a mass flow rate were identified by tank test simulation. The parameter values of CPM were selected default condition on atmosphere space and the best conditions on Table 1 to reproduce the gas flow. The effect of gas flow reproduction in atmosphere space was examined to airbag deployment.
RESULTS

Visualization Experiments and Simulation of Gas Flow

The gas flow from inflator was clearly visualized as a dark image in schlieren method. The results of visualization are shown in Figure 3. The large distortion parts of parallel light were removed and observed as shadow. Since the inflator gas had a high temperature and a high pressure, the gas produced a distinct distortion that was different from the atmosphere. In these results, the difference of the flow with or without retainer was clearly observed. The gas trended to be released radially and vertically from orifices on the inflator for without a retainer. On the other hands, for with a retainer, the flow along the wall of retainer was observed. Additionally, the flow along the wall did not diffuse immediately after release.

![Figure 3. Visualization of inflator gas by schlieren method (Left: without retainer, Right: with retainer).](image)

The gas flow of simulation by CPM for without the retainer in atmosphere space trended to be diffused randomly. The behavior of particles did not show radial flow. (See Figure 4A) To reproduce this trend, when the cone angle parameter in Table 1 was set appropriately, radial flow was shown in Figure 4B.

On the other hands, for with retainer, the gas flow of simulation did not reproduce the experimental result. (See Figure 4C) The parameters from No. 3 to No. 7 in Table 1 did not affect to gas flow behavior. (See Figure 4D) For with retainer, when jet direction was set as axial jet along the wall of retainer, the diffusion range of flow was slightly narrowed, however a directional flow was not reproduced. (See Figure 4E) When additional parameter cone angle was set appropriately, the flow was reproduced as same as the experimental result. (See Figure 4F)
Deployment Experiments and Simulation of Driver Side Airbag

Comparison between deployment experiment and simulation of driver side airbag was conducted. When the jet direction was set radially and vertically of the inflator in simulation, the result shows that the deployment behavior was delay against experiment. Approximately half force of the experimental result occurred in retainer fixed points. (See Figure 5A)

When the jet direction was set as axial and along the wall of retainer and cone angle was set appropriately, the deployment time and deployment force were almost reproduced the experimental results. (See Figure 5B)
DISCUSSION

According to the result comparison between experiment and simulation, when the particles were released to relatively wide space, random movement of each particle is dominant, the gas flow was found to be not sufficiently reproducible by CPM.

The actual gas flow from orifices on the inflator is released in vertical direction on orifices. When the gas flow from inflator with retainer is released, if it is assumed that the gas flow outlet faces open side of the retainer, the gas is released perpendicular to the open side of the retainer.

In CPM, because the particles diffuse randomly from orifices of the inflator, it should apply correction function such as cone angle.

CONCLUSIONS

The visualization experiment and simulation of the gas flow from inflator provide the following findings.

Using schlieren method, the gas flow to atmosphere space is radially released perpendicular to orifices on the inflator. When the retainer is installed on inflator, the gas flow along wall of retainer is produced and the gas flow is perpendicular to open side of the inflator.

In simulation of CPM, particles from inflator behave randomly. To reproduce the actual radial flow, it should have a correction function, such as a cone angle. When the gas flow with retainer is reproduced, it should set the jet direction from open side of retainer and cone angle.

Applying the above conditions to deployment simulation of driver side airbag, the deployment behavior and the deployment force property are reproduced the experimental results.

Figure 5. Comparison of deployment behavior & deployment force between experiment and simulation (Left: Radial jet, Right: Axial jet with Cone angle).
ACKNOWLEDGEMENTS

We wish to acknowledge valuable experiments and discussions with honorary Prof. Takayama at Institute of Fluid Science, Tohoku University.

REFERENCES