Crashworthiness of Front Side Members in an Auto-body Considering the Fabrication Effect

H. Huh*, K. P. Kim, S. H. Kim
Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology
Science Town, Daejeon, 305-701, Korea
*E-mail: hhuh@kaist.ac.kr

H. S. Kim, S. K. Hong
Passenger Car Body Engineering Team, Kia Motors Corporation
Kwangmyung, Kyungki-do, 423-701, Korea

Abstract

This paper is concerned with crash analysis for a front side member of an auto-body considering the effect of fabrication. The front side member is fabricated with sheet metal forming processes that induce forming histories such as the plastic work hardening and non-uniform thickness distribution. Numerical simulation is carried out with LS-DYNA3D in order to identify forming effects on the crashworthiness. The crash analysis of the front side member with the forming effect leads to a different result from that without the forming effect. Crashworthiness factors such as the load-carrying capacity, the crash mode and the energy absorption are calculated to investigate and identify forming effects. It is fully demonstrated that the design of auto-body members needs to consider the forming effects for accurate assessment of the load-carrying capacity and the deformation mechanism of the formed members.

Key-Words: Crashworthiness, Crash analysis, Front side member, Forming effect

1. Introduction

The crashworthiness of a car has to be evaluated with the load-carrying capacity and the crash mode at the initial stage of auto-body design. Auto-body members such as a front side member should be designed to efficiently absorb the kinetic energy during the car crash in order to secure occupants from the impact and penetration. The estimation of the energy absorption efficiency of auto-body members requires the accurate crash analysis for the load-carrying capacity and the crash mode. In order to accomplish reliable crash simulation, crashworthiness of auto-body members should be evaluated considering the effect of stamping and forming as well as the dynamic properties of materials. As most load-carrying members of an auto-body are fabricated from the sheet metal forming process, they could possess wrinkling and thinning induced from forming as well as non-uniform distributions of the effective plastic strain and the thickness strain according to the forming condition and their final shapes. Many crash analyses have been, however, carried out neglecting the forming effect induced by stamping and forming processes for estimation of the crashworthiness of an auto-body, providing erroneous results in the crash mode and the amount of crash. Recently, the crash analysis has been performed for auto-body members considering forming effect such as the strain hardening and the non-uniform thickness distribution[1-6]. These studies insisted that the crash analysis of auto-body structures should be carried out considering forming effects for the purpose of reliable assessment.

In this paper, the forming histories of a front side member are obtained from simulation of stamping and forming processes so that they can be considered in estimation of its crashworthiness. Since front side members should play an important role in absorption of the kinetic energy during the front crash, they are fabricated from sheet metals. Forming analysis of each panel of the front side member is carried out with an explicit elasto-plastic finite element analysis code, LS-DYNA3D. Non-uniform distributions of the effective plastic strain and the thickness strain in formed panels are obtained as the forming history from simulation of sheet metal forming. As the first step, drawbead analysis is performed for calculation of the restraining force of drawbeads with an implicit elasto-plastic finite element analysis code, ABAQUS/Standard. Secondly, the calculated restraining force is applied to forming simulation of each panel as the equivalent restraining force on the flange region. The thickness strain of the inner panel, frame-frt-in, is compared with the thickness strain in a real product for verification of the analysis result.

Crash analysis of the front side member is carried out imposing the forming analysis result on the initial condition. Numerical simulation is performed with LS-DYNA3D in order to evaluate the crashworthiness of the front side member. In order to consider the non-uniform distributions of the effective plastic strain and the thickness as the condition for crash analysis, the forming histories are mapped into the new finite element mesh system. The crash analysis results of the front side member considering the forming histories are compared with that without the forming effect. The forming effect on the crashworthiness is investigated for non-uniform distribution of the thickness and the effective plastic strain separately. The crash analysis results well demonstrate
that these forming histories greatly change the crash mode, the load-carrying capacity and the energy absorption efficiency of the front side member. It is noted from the results that design of auto-body members needs to consider the forming effects for a proper and accurate evaluation of the crashworthiness of a car with fabricated members.

2. Forming analysis of the front side member

A front side member is composed of seven panels: frame-frt-in; frame-frt-out-A; frame-frt-out-B; reinf-frt-frame-A; reinf-frt-frame-C; reinf-frt-frame-D; and hook-tie-down. The forming histories are calculated with the direct forming analysis for four panels that have great influences on the behavior of the front crash. For the sake of the computational efficiency, the restraining forces of drawbeads in the dies are calculated with an implicit elasto-plastic finite element code, ABAQUS/Standard. The forming analysis is then carried out for four panels, imposing the calculated restraining forces as boundary conditions of the equivalent drawbead[7-10]. The forming analysis, which is made up of the binder wrap process and the punch forming process, is performed with explicit elasto-plastic finite element code, LS-DYNA3D. Calculated forming results are applied to the crash simulation as the initial condition.

2.1 Drawbead analysis

The punch and die profiles for the finite element analysis of the drawbead forming process are extracted from the geometric data of the tool set as shown in Figure 1 for frame-frt-in. Drawbeads are employed with the different shape and size according to each panel. The binder wrap analysis as well as the drawing analysis of the drawbead is performed to calculate the restraining force of drawbeads for accurate forming simulation. Calculated equivalent drawbead forces are applied to the forming analysis of the front side member. Figure 2 explains geometric shapes of the circular and rectangular drawbeads for the simulation. The dimensions of circular drawbeads employed in forming of four panels are explained in Table 1. The dimension of the rectangular drawbead in the reinf-frt-frame-A is \( H = 6 \text{ mm}, L = 7.65 \text{ mm}, r_1 = 2.18 \text{ mm} \) and \( r_2 = 1.5 \text{ mm} \). Table 2 shows the initial sheet thickness and material properties for forming simulation of four panels. The finite element model of an initial blank sheet has four layers of finite elements for accurate description of bending deformation.

Table 1. Dimensions of circular drawbeads in the front side member (unit = mm).

<table>
<thead>
<tr>
<th>Part</th>
<th>Bead type</th>
<th>( R )</th>
<th>( r )</th>
<th>( H )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frame_frt_in</td>
<td>Circular bead</td>
<td>4.8</td>
<td>6.2</td>
<td>3.0</td>
</tr>
<tr>
<td>Frame_frt_out_A</td>
<td>Circular bead</td>
<td>4.0</td>
<td>5.2</td>
<td>6.0</td>
</tr>
<tr>
<td>Reinf_frt_frame_A</td>
<td>Circular bead 1</td>
<td>4.5</td>
<td>3.8</td>
<td>3.2</td>
</tr>
<tr>
<td>Reinf_frt_frame_A</td>
<td>Circular bead 2</td>
<td>4.5</td>
<td>3.8</td>
<td>5.2</td>
</tr>
<tr>
<td>Reinf_frt_frame_C</td>
<td>Circular bead 1</td>
<td>6.5</td>
<td>4.7</td>
<td>3.0</td>
</tr>
<tr>
<td>Reinf_frt_frame_C</td>
<td>Circular bead 2</td>
<td>6.5</td>
<td>4.7</td>
<td>5.0</td>
</tr>
<tr>
<td>Reinf_frt_frame_C</td>
<td>Circular bead 3</td>
<td>6.5</td>
<td>4.7</td>
<td>6.0</td>
</tr>
</tbody>
</table>

Table 2. Material properties and initial thickness of panels in the front side member.

<table>
<thead>
<tr>
<th>Part (Material)</th>
<th>Sheet thickness and flow stress curve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frame_frt_in (SPRC40)</td>
<td>Initial sheet thickness: 1.6 mm ( \bar{\sigma} = 734.7(0.01 + \varepsilon)^{0.245} ) (MPa)</td>
</tr>
<tr>
<td>Frame_frt_out_A (SPRC40)</td>
<td>Initial sheet thickness: 1.2 mm ( \bar{\sigma} = 734.7(0.01 + \varepsilon)^{0.245} ) (MPa)</td>
</tr>
<tr>
<td>Reinf_frt_frame_A (SAPH38)</td>
<td>Initial sheet thickness: 0.9 mm ( \bar{\sigma} = 768.5(0.023 + \varepsilon)^{0.265} ) (MPa)</td>
</tr>
<tr>
<td>Reinf_frt_frame_C (SPRC45)</td>
<td>Initial sheet thickness: 2.0 mm ( \bar{\sigma} = 823.8(0.012 + \varepsilon)^{0.275} ) (MPa)</td>
</tr>
</tbody>
</table>

Table 3 shows restraining forces that are obtained at the steady state for each drawbead as the equivalent forces in forming simulation. A rectangular drawbead is employed in forming of the reinf-frt-frame-A in order to prevent the material flow of the sheet from drawing in with the large restraining force.

Table 3. Restraining forces of drawbeads in the front side member.

<table>
<thead>
<tr>
<th>Part</th>
<th>Bead type</th>
<th>Restraining force</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frame_frt_in</td>
<td>Circular bead</td>
<td>226 N/mm</td>
</tr>
<tr>
<td>Frame_frt_out_A</td>
<td>Circular bead</td>
<td>243 N/mm</td>
</tr>
<tr>
<td>Reinf_frt_frame_A</td>
<td>Circular bead 1</td>
<td>246 N/mm</td>
</tr>
<tr>
<td>Reinf_frt_frame_A</td>
<td>Circular bead 2</td>
<td>271 N/mm</td>
</tr>
<tr>
<td>Reinf_frt_frame_A</td>
<td>Rectangular bead</td>
<td>294 N/mm</td>
</tr>
<tr>
<td>Reinf_frt_frame_C</td>
<td>Circular bead 1</td>
<td>217 N/mm</td>
</tr>
<tr>
<td>Reinf_frt_frame_C</td>
<td>Circular bead 2</td>
<td>235 N/mm</td>
</tr>
<tr>
<td>Reinf_frt_frame_C</td>
<td>Circular bead 3</td>
<td>278 N/mm</td>
</tr>
</tbody>
</table>
2.2 Forming analysis

Direct forming analyses have been carried out for four panels of the front side member. A commercial explicit finite element code, LS-DYNA3D is employed in forming simulation for computational efficiency since it requires tremendous calculation time to simulate forming processes with the implicit finite element method. The initial sheet thickness and material properties were shown in the Table 2. The coulomb friction coefficient is 0.15 between the sheet and tools. The blank holding force of 100 kN are applied to the blank holder. The binder wrap analysis and the punch forming analysis are carried out with calculated equivalent drawbead forces listed in Table 3 as boundary conditions.

The die, the punch and the blank holder are modeled with finite element patches for forming simulation of each panel. Figure 3 shows the tooling system for the analysis of the frame-frt-in. Drawbeads are expressed with curved lines and attached to the blank holder as shown in Figure 1. An r-type adaptive mesh system is adopted for the precise description of the formed shape. Figure 4 shows the effective plastic strain distribution and the thickness distribution in the frame-frt-in after forming simulation. The thickness distribution calculated has been compared to that in the real part for the frame-frt-in. Figure 5 shows the quantitative comparison of the thickness variation along the designated line between the real product and the finite element simulation result. The figures show very close coincidence between the measured results and the calculated results. The comparison fully demonstrates that the result from finite element forming simulation of the front frame member is highly reliable and can be applied to the crash analysis as the initial condition without distorting real crash behavior.

![Figure 3. Initial setting of tools and the blank for the numerical analysis of the frame-frt-in in the front side member.](image)

![Figure 4. Final forming results of the frame-frt-in: (a) distribution of the effective plastic strain; (b) distribution of the thickness strain.](image)

![Figure 5. Comparison of thickness distribution of the frame-frt-in: (a) measured section; (b) section A-A'; (c) section B-B'; (d) section C-C'.](image)

Forming analyses of other panels were also carried out with the same procedures in order to obtain the effective plastic strain distribution and thickness distribution as the forming history that is to be considered as forming effects in the crash simulation. Distributions of the effective plastic strain and the thickness were mapped into new finite element mesh systems for the crash simulation.

3. Crash analysis of the front side member

A finite element model of the whole front side member for the crash analysis is shown in Fig. 6. The total mesh system consists of 29842 four-node shell elements and 30788 nodal points. Finite element models of seven panels that compose the front side member are illustrated in Figure 7 with the number of nodes and elements specified for each panel. The panels are obtained from the formed ones by trimming. Forming histories obtained from the forming analysis in the previous section are utilized as the initial condition of the crash analysis for accurate assessment of the crashworthiness.
The material specification and the thickness for seven panels of the front side member are shown in Table 4. Dynamic behavior of materials is described with the Johnson-Cook constitutive relation as shown in Equation (1), where $A$, $B$, $n$, $C$, $m$ are material constants [11, 12].

$$
\bar{\sigma} = \left[ A + B \bar{\varepsilon} \right] \left[ 1 + C \ln \frac{\bar{\varepsilon}}{\bar{\varepsilon}_0} \right] \left[ 1 - T^m \right]
$$

(1)

where

$$
T = \frac{T - T_{room}}{T_{melt} - T_{room}}, \quad \bar{\varepsilon}_0 = 1 \text{ sec}.
$$

(2)

The constants for materials of the front side member are given in Table 5.

Boundary conditions for the crash analysis of the front side member are explained in Figure 8. One end of the front side member is fixed and the other end crashes by a moving rigid wall. The mass of the rigid wall is 200 kg and the initial velocity is 13.3 m/s.

Crash analysis of the front side member is carried out considering the forming histories obtained from the forming analysis. The analysis result is compared with the one without forming effects. The thickness distribution and the effective plastic strain distribution are considered separately and all together as the initial condition of the crash analysis of the front side member. The crash analysis has been carried out for 30 ms.

Figure 9 shows deformed shapes of the front side member for four different cases of simulation: the first one is without the forming effect; the second one is with the thickness distribution; the third one is with the effective plastic strain distribution; and the fourth one is with all forming histories. The deformation proceeds from the struck end since the end region has several grooves to induce axial folding. After the front region is crushed with folding, the deformation proceeds to the rear region absorbing more kinetic energy. The results show that when the effective plastic strain is considered as the forming histories, the middle region deforms more than others. When the thickness is considered, deformation is concentrated on dimples in the grooved region and thus the energy is absorbed less than others. Figure 10 demonstrates that the deformed shape with all forming effects considered is different from the one of the designed front side members that did not consider any forming effect. The region with dimples undergoes severer deformation when all forming effects are considered than when the forming effect is not considered.
Figure 9. Deformed shapes of the front side member at 30 msec: (a) without forming effect; (b) with the thickness distribution; (c) with the effective plastic strain distribution; (d) with all forming histories.

Figure 10. Comparison of deformed shapes of the front side member: (a) without forming effects; (b) with all forming effects.

The mid region, however, undergoes more remarkable bending when the forming effect is not considered than the one with all forming effects considered. These results show that the deformation mode is greatly changed when forming histories are considered in the crash analysis. The most influencing factor is the non-uniform distributions of the effective plastic strain and the thickness. The comparison fully demonstrates that the forming histories have to be considered in the crash analysis for accurate assessment of the crashworthiness.

The reaction forces normal to the wall are plotted with respect to the crushing distance in Figure 11. The curves were obtained by filtering the original curves from the crash analysis with the SAE600 filter. The initial peak load increases by 13 % when only the effective plastic strain is considered and decreases slightly when only the non-uniform thickness is considered. The reaction force tends to increase more with deformation in case of the analysis considering the effective plastic strain than without considering the effective plastic strain. The results explain that the strain hardening with the effective plastic strain from forming processes makes the maximum load increased and the non-uniform thickness distribution with the thinned region can be considered as the initial defect of the front side member.

The absorbed energy during deformation is plotted in Figure 12. The figure also demonstrates that energy absorption increases remarkably when the effective plastic strain is considered and decreases slightly when the non-uniform thickness distribution is considered. The energy absorption of the front side member has a larger value when all forming effects are considered than the one without forming effect. The difference is 5.3 % at the crushing distance of 100 mm, 10.2% when the crushing distance is 200 mm, and 17.3 % when the crushing distance is 250 mm. The most important phenomenon is that the energy absorption rate increases with deformation when the effective plastic strain is considered in the analysis, while the energy absorption rate decreases with deformation when the effective plastic strain is not.
considered. These results demonstrate that the strain hardening resulted from forming processes is dominant in calculation of the reaction force and energy absorption. It is noted from the results that the crash analysis of the front side member has to be carried out considering the forming effect, especially the effective plastic strain.

4. Conclusion

Crash analysis of a front side member has been carried out considering the forming effect in order to evaluate the crashworthiness investigating the crash mode, the reaction force and the energy absorption. The forming analysis of the front side member has been carried out to obtain the distribution of the thickness and the effective plastic strain that was considered as the initial condition of the crash analysis. The crash analysis results considering the forming effect were compared with that without forming effects. The comparison demonstrates that the effective plastic strain is dominant in calculation of the crash mode, the reaction force and the energy absorption due to the strain hardening. It is fully demonstrated that the forming history should be considered for accurate assessment of the crashworthiness at the design stage of auto-body members.

References


