Vehicle Structural Collapse Analysis using a Finite Element Limit Method

H. S. Kim and H. Huh*

Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology
373-1 Kusong-dong Yusong-gu, Taejon 305-701, Korea

Abstract This paper discusses the collapse behavior of vehicle structures under quasi-static loading conditions. The finite element limit analysis for three-dimensional structures is formulated based on the duality theorem in plasticity. The analysis considers sequential deformation of structures with work-hardening effects. The collapse analysis for a S-shaped frame is conducted using both the limit analysis program developed here and the commercial code ABAQUS. Results show a good agreement in load-carrying capacity and deformation mode predictions. Example studies also demonstrate that the method presented can be used to identify the weak part of a structure and change its design to enhance the load-carrying capacity and reduce vehicle passenger compartment distortions.

Keywords: 3-D finite element limit analysis, collapse analysis, work-hardening, space-framed vehicle structure, S-shaped frame

* : Corresponding author

Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology
373-1 Kusong-dong Yusong-gu, Taejon 305-701, Korea
Vehicle Structural Collapse Analysis using a Finite Element Limit Method

H. S. Kim and H. Huh

Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology
373-1 Kusong-dong Yusong-gu, Taejon 305-701, Korea

Abstract  This paper discusses the collapse behavior of vehicle structures under quasi-static loading conditions. The finite element limit analysis for three-dimensional structures is formulated based on the duality theorem in plasticity. The analysis considers sequential deformation of structures with work-hardening effects. The collapse analysis for a S-shaped frame is conducted using both the limit analysis program developed here and the commercial code ABAQUS. Results show a good agreement in load-carrying capacity and deformation mode predictions. Example studies also demonstrate that the method presented can be used to identify the weak part of a structure and change its design to enhance the load-carrying capacity and reduce vehicle passenger compartment distortions.

Keywords : 3-D finite element limit analysis, collapse analysis, work-hardening, space-framed vehicle structure, S-shaped frame

INTRODUCTION

Vehicle structures should be designed to be able to efficiently absorb kinetic energy for good crashworthiness properties. To improve this property, the vehicle load-carrying capacity and collapse mode have to be estimated based on simple models at the initial design stage. Using the finite element limit analysis at the initial design stage, vehicle development time and cost can be reduced by providing sufficient structural strength and
crushable space to protect passengers. Lyle (1991) demonstrated the application of appropriate crash analysis to vehicle concept design.

In predicting the gross response of a vehicle crash, the plastic hinge technique is often used with the concept of limit analysis. The plastic hinge technique assumes that all plastic deformation occurs at the prelocated hinges. Chang (1974) developed the design-analysis method based on the principle of limit analysis for estimating the ultimate load-carrying capacity and associated collapse mechanism of a passenger compartment. He experimentally obtained the stiffness data for major vehicle body connections, and used these data in a baseline mathematical model. And then he studied the influence of each connection on structural responses under four representative loading conditions. McIvor et al. (1977) proposed a structural theory for the large static plastic deformation of space frames. The analysis assumed that the plastic deformation was confined to the idealized hinges located at node points. Nikravesh and Chung (1984) employed a plastic hinge concept and divided the structure into several components connected by the plastic hinge, which was modeled by a joint-spring combination. Kang (1996) developed a simple finite element model with nonlinear spring elements to represent the plastic hinge behavior at weak regions and predicted the load–displacement curve and roof crush resistance from the analysis. To use the plastic hinge technique, however, the structural characteristics and appropriate hinge locations need to be determined in advance from either experiments or semi-empirical equations. The quality of the solution depends on these characteristics and hinge locations.

The limit analysis is an efficient concept to estimate the load-carrying capacity and collapse mechanism of structures. When combined with the finite element method, the limit analysis has been in remarkable progress. Huh and Yang (1991) solved the plane stress problem based on the duality relation. Yang (1993) used the sequential limit analysis to compute the large deformation of trusses and frames. Because the method involves only geometric updating, accurate updating can be achieved with relatively large step sizes. In an incremental analysis, however, complicated stress updating is needed. Huh and Lee (1993) also suggested to consider the work-hardening effect in the finite element limit analysis. Liu et al. (1995) formulated the three-dimensional limit analysis as a mathematical problem and conducted the limit analysis for perfect plastic structures. However, in Liu’s study the work-hardening effect was not included and only the initial limit load was calculated. Huh et al. (1998) derived a three-dimensional limit formulation with work-hardening material properties as a general algorithm for limit solutions. In the work presented here, S-shaped frame was chosen as an example to compare the limit analysis with the elasto-plastic analysis.

To estimate the energy absorption efficiency and collapse mechanism of vehicle structures, collapse
analysis is needed to study the load-carrying capacity and collapse mode with respect to various loading conditions. Mahmood and Paluszny (1981) considered a semi-empirical approach to develop a design method for sheet metal columns under crash loadings. The effect of collapse mode on energy absorption depends on loading conditions, structural shapes, and material properties. Magee and Thornton (1978) analyzed the energy absorption efficiency of structures with respect to changes of material properties and structural shapes. Vehicle design should aim at ensuring the crashworthiness by developing the structures to efficiently absorb crash energies. The major deformed parts and their deformation modes can be studied using a collapse analysis. The vehicle crashworthiness should be assured at the initial stage of a conceptual design.

In this study, the collapse behavior of a simplified vehicle model under quasi-static loading conditions is investigated using finite element limit analysis. The load-carrying capacity of structures decreases after initial collapse since the structures become weaker after progressive deformations. Therefore, the initial collapse load is an important factor in vehicle crashworthiness. The objective of this work is to evaluate the ability to improve vehicle design based on the result from collapse studies using finite element limit analysis. The finite element limit analysis can be used to predict plastic collapse loads and collapse modes systematically. Therefore, it is a useful tool to examine the safety of a vehicle structure at its initial design stage.

LIMIT ANALYSIS THEORY

The limit analysis formulation consists of the primal and dual formulations. It addresses plastic materials which obey the convex yield criterion and the associated flow rule. The primal formulation can be derived from the statically and constitutively admissible conditions in the form of a constrained maximization problem, i.e.,

\[
\begin{align*}
\text{maximize} & \quad q(\sigma) \\
\text{subject to} & \quad \nabla \cdot \sigma = 0 \quad \text{in } \Omega \\
& \quad \sigma \cdot \mathbf{n} = q \mathbf{t} \quad \text{on } \partial \Omega_S \\
& \quad \|\sigma\| \leq \sigma_0 \quad \text{in } \Omega
\end{align*}
\]

(1)

where \( \sigma \) is the stress tensor in the reference domain \( \Omega \), \( \mathbf{t} \) is a traction force vector on the boundary surface \( \partial \Omega_S \) whose unit outer normal vector is \( \mathbf{n} \), and \( q \) is a positive real parameter of proportional loading. The statically admissible set in the stress space can be formed with the state of stress that satisfies the equation of
equilibrium and the static boundary conditions. The von-Mises yield criterion $\|\sigma\|_{(V)} = \sigma_a$ is regarded as the constitutively admissible set. Although $q$ can be uniquely obtained in Eqn. (1), $\sigma$ may or may not be unique.

Eqn. (1) defines a convex surface in the function space $\mathbb{R}^{3\times 3}(D)$ and seeks the maximum of the positive scaling factor $q(\sigma)$, while the magnitude of the stress matrix, $\sigma$, is constrained by the von-Mises yield condition in the convex norm. While this kind of maximization problem, the lower bound formulation in plasticity, can be solved by finite dimensional approximation, it is not practical and efficient since the solution needs to be determined in a stress space.

The convex problem has a dual one which corresponds to the upper bound formulation. The minimum solution of dual formulation is equal to the maximum $q(\sigma)$ in Eqn. (1). To construct the dual problem, the principle of virtual work is used to form a weak equilibrium equation,

$$\int_D u : (\nabla \cdot \sigma) \, d\Omega = 0 \quad , \forall u$$

(2)

where $u$ is an arbitrary function in $\mathbb{R}^3(D)$ with the physical interpretation as an admissible velocity function. The above integration is carried out in the reference domain of two or three dimension for each incremental step. The admissible $u$, which satisfies the kinematic boundary conditions on $\partial D_k$, will lead to the equivalent variational statement, by applying the divergence theorem and static boundary conditions,

$$\int_D \sigma : \varepsilon \, d\Omega = q \int_{\partial D_k} t \cdot u \, d\Gamma$$

(3)

where $\varepsilon$ is the strain rate matrix and the symbol $: \,$ denotes the inner product operator between two matrices. Eqn. (3) can be restated in an alternative way as follows

$$q(\sigma) = \int_D \sigma : \varepsilon \, d\Omega = \int_D \sigma \varepsilon_{ij} \, d\Omega$$

(4)

where the boundary integral in Eqn. (3) has been normalized, i.e.,
\begin{equation}
\int_{\partial \Omega} t \cdot u \, d\Gamma = 1
\end{equation}

Using the principle of maximum work dissipation or by a generalized Hölder inequality, the term \( \sigma : \varepsilon \) can be rewritten as follows,

\begin{equation}
\sigma : \varepsilon = |\sigma : \varepsilon| \leq \|\sigma\|_{(V)} \|\varepsilon\|_{(-V)} = \bar{\sigma} \bar{\varepsilon}
\end{equation}

where \( \|\sigma\|_{(V)} \) denotes the von-Mises norm of the stress, and \( \|\varepsilon\|_{(-V)} \) denotes the minus von-Mises norm of the strain rate, which define the equivalent stress and strain rate, respectively.

The inequality is sharp, i.e., the equality holds when \( \varepsilon \) is chosen to be proportional to the gradient of the yield function. The sharpness condition

\begin{equation}
\varepsilon = k \nabla \|\sigma\|_{(V)}
\end{equation}

is the well known normality condition in plasticity, where \( k \) is a proportional factor. Consequently, an upper bound to the functional, \( q(\sigma) \), can be established through the sequence of inequalities as

\begin{equation}
q(\sigma) = \int_{\Omega} \sigma : \varepsilon \, d\Omega \\
\leq \int_{\Omega} |\sigma|_{(V)} \|\varepsilon\|_{(-V)} \, d\Omega \\
\leq \sigma_o \int_{\Omega} \|\varepsilon\|_{(-V)} \, d\Omega \\
= \tilde{q}(u)
\end{equation}

where the upper bound functional \( \tilde{q}(u) \) depends only on the kinematically admissible function \( u \). Based on the inequality in Eqn. (8) and the existence of the absolute minimum of \( \tilde{q}(u) \), the dual formulation may be stated as
minimize $\tilde{q}(u)$  
subject to $\tilde{q} = \sigma_o \int_{\Gamma} \| \epsilon \| \, d\Omega$  
\[ \int_{\Gamma} \tau : u \, d\Gamma = 1 \nonumber \]  
\[ \text{Tr}(\epsilon) = 0 \nonumber \]  
Kinematic boundary conditions

The minimum solution of $\tilde{q}(u)$ is equal to the maximum $q(\sigma)$ in formulation (1), within the smallest part of the kinematically admissible function spaces, by the duality relation (Huh and Yang, 1991). In real problems, a general solution of formulation (9) could be obtained using numerical methods.

In the calculation for work-hardening materials, the effective stress–strain curve is considered as step-wise constant, but the magnitude of the current yield stress, $\sigma_o$, is adjusted based on the effective strain by successive iterations using the bisection method (Huh and Lee, 1993, 1998), i.e.,

\[ \sigma_o = \sigma = F(W_p) \text{ or } H(\bar{\varepsilon}^p) \quad (10) \]

where $F(W_p)$ and $H(\bar{\varepsilon}^p)$ represent functions for work-hardening and strain-hardening, respectively, and $\bar{\varepsilon}^p$ indicates the equivalent plastic strain. Considering the current yield stress in each incremental step, there is no need to check whether the stress–strain relation is correctly tracking the given data due to the nature of the formulation. The above concept extended from the conventional limit analysis makes it possible to simulate the collapse behavior of three-dimensional structures with work-hardening materials by using finite element limit analysis. Although there might be a small amount of error with the assumption that the current yield stress is considered as a constant, it ensures stable convergence and computational efficiency even in shakedown. It also removes the accumulated error that often comes from the calculation of plastic or elasto-plastic tangent modulus in each incremental step. The amount of increment should allow that the maximum effective strain in each step is less than 0.2%. The current yield stress can be obtained from a typical uniaxial stress–strain relation

\[ \sigma = \sigma_o (1 + A \bar{\varepsilon}^p)^n \quad (11) \]

where $A$ and $n$ are the constants for a given material, and $\sigma_o$ is the initial yield stress.
FINITE ELEMENT PROCEDURES AND MINIMIZATION TECHNIQUE

The dual formulation is discretized into the sub-domains of finite elements and reduced to a convex problem in the finite dimensional space $\mathbb{R}^N$, where $N$ is the total number of discrete variables. To guarantee the incompressibility condition, the objective functional is modified as

$$\bar{q}(\mathbf{u}) = \bar{\sigma} \int_D \mathbf{e} : \mathbf{e} \, dV_D + \Lambda \int_D u^T \, dV_D$$

where $\Lambda$ is a penalty factor which can be a large number.

For the finite dimensional approximation, the strain rate vector in a three-dimensional space can be expressed in a matrix form as

$$\mathbf{e} = \begin{pmatrix} \varepsilon_x & \varepsilon_y & \sqrt{2}\varepsilon_{xy} & \sqrt{2}\varepsilon_{yz} & \sqrt{2}\varepsilon_{xz} \end{pmatrix}^T = \mathbf{B} \mathbf{U}$$

where $\mathbf{B}$ is the gradient matrix related to the strain rate components and $\mathbf{U}$ is the nodal velocity vector. Then, the effective strain rate can be written as

$$\bar{\mathbf{e}} = \frac{2}{3} \varepsilon_i \varepsilon_i = \frac{2}{3} \mathbf{U}^T \mathbf{B}^T \mathbf{BU} = \sqrt{\mathbf{U}^T \mathbf{K}_v \mathbf{U}}$$

where $\mathbf{K}_v^2 = \frac{2}{3} \mathbf{B}^T \mathbf{B}$. The matrix form of volumetric strain rate can be written as

$$\varepsilon_v = \varepsilon_x + \varepsilon_y + \varepsilon_z$$

$$= \begin{bmatrix} 1 & 1 & 0 & 0 \end{bmatrix} \begin{pmatrix} \varepsilon_x & \varepsilon_y & \sqrt{2}\varepsilon_{xy} & \sqrt{2}\varepsilon_{yz} & \sqrt{2}\varepsilon_{xz} \end{pmatrix}^T = \mathbf{A} \mathbf{B} \mathbf{U} = \mathbf{K}_v^2 \mathbf{U}$$
where $A$ is a vector, $\{111000\}$ and $K^T_+ = AB$.

The objective functional is approximated using finite elements in a quadratic form of the discrete vector representation of the velocity field, $U$, i.e.,

$$\tilde{q}(U) = \sum_{e=1}^{E} \left[ U^T K^+_1 U + U^T K^+_2 U \right]$$

where the integer $E$ is the total number of the finite elements, and

$$K^+_1 e = \int_{\Omega} \frac{K^+_1 e}{U^T K^+_1 U} \, d\Omega$$

$$K^+_2 e = \Lambda \int_{\Omega} K^+_2 e \, d\Omega$$

The element stiffness matrix in Eqn. (17) can be approximated as follows for an iterative scheme

$$\left( K^+_1 \right)_{n} = \frac{\int_{D_e} \left( K^+_1 \right)_{n}}{\left( U^T K^+_1 U \right)_{n-1}} \, d\Omega$$

where $(n-1)$ and $(n)$ indicate the previous iterative step and current step, respectively.

The approximation of the dual formulation (9) becomes a constrained quadratic problem, i.e.,

$$\text{minimize } \tilde{q}(U) = U^T K U$$

subject to $C^T U = 1$ 

where $C^T U = 1$ represents the normality condition and the kinematic boundary conditions are absorbed into the matrix $K$ and vector $C$ which will yield the optimum $U_n$ and the minimum value $q_n$ associated for each iterative step. The constrained minimization formulation (19) is converted to an unconstrained problem in the
solution procedure with the use of the Lagrange multiplier method, i.e.,

\[
\text{minimize } \Phi(U) = U^T K U - 2 \lambda (C^T U - 1) \tag{20}
\]

where \( \lambda \) is the Lagrange multiplier.

Differentiating Eqn. (20) with respect to the displacement and Lagrange multiplier results in the following equations:

\[
\frac{\partial \Phi}{\partial U} = 2KU - 2\lambda C = 0 \tag{21}
\]

\[
\frac{\partial \Phi}{\partial \lambda} = C^T U - 1 = 0 \tag{22}
\]

From Eqn. (21) and (22), following equation can be obtained

\[
U = \lambda K^{-1} C \tag{23}
\]

\[
\lambda = U^T K U \tag{24}
\]

or

\[
\lambda = \frac{1}{C^T K^{-1} C} \tag{25}
\]

When the displacement boundary condition is given, the displacement vector can be divided into two parts as

\[
U = \begin{bmatrix} U_1 \\ U_2 \end{bmatrix}
\]

where \( \{U_1\} \) is an unknown displacement vector and \( \{U_2\} \) is known from the boundary condition. The stiffness matrix \( K \) also can be divided into sub-matrices corresponding to the given displacement boundary
condition as

\[
K = \begin{bmatrix}
K_{11} & K_{12} \\
K_{21} & K_{22}
\end{bmatrix}
\]  \hspace{1cm} (27)

The load vector also can be divided into two parts as

\[
C = \begin{bmatrix}
C_1 \\
C_2
\end{bmatrix}
\]  \hspace{1cm} (28)

where \(C_1\) becomes zero when only the displacement boundary condition is given and \(C_2\) is unknown at the region where the displacement boundary condition is given. Using the partitioned stiffness matrices and displacement vectors, Eqn. (21) can be restated as

\[
\begin{bmatrix}
K_{11} & K_{12} \\
K_{21} & K_{22}
\end{bmatrix}
\begin{bmatrix}
U_1 \\
U_2
\end{bmatrix} = \lambda\begin{bmatrix}
C_1 \\
C_2
\end{bmatrix}
\]  \hspace{1cm} (29)

Eqn. (29) can be divided into two independent sets of equations as

\[
\begin{bmatrix}
K_{11}
\end{bmatrix}\begin{bmatrix}
U_1
\end{bmatrix} + \begin{bmatrix}
K_{12}
\end{bmatrix}\begin{bmatrix}
U_2
\end{bmatrix} = \lambda\begin{bmatrix}
C_1
\end{bmatrix} = \{0\}
\]  \hspace{1cm} (30)

\[
\begin{bmatrix}
K_{21}
\end{bmatrix}\begin{bmatrix}
U_1
\end{bmatrix} + \begin{bmatrix}
K_{22}
\end{bmatrix}\begin{bmatrix}
U_2
\end{bmatrix} = \lambda\begin{bmatrix}
C_2
\end{bmatrix}
\]  \hspace{1cm} (31)

From the Eqn. (30), \(\{U_1\}\) can be obtained as

\[
\{U_1\} = -\begin{bmatrix}
K_{11}
\end{bmatrix}^{-1}\left(\begin{bmatrix}
K_{12}
\end{bmatrix}\begin{bmatrix}
U_2
\end{bmatrix}\right)
\]  \hspace{1cm} (32)

and then the displacement vector is restated as
Using the vector \( \{ U_1 \} \) obtained from Eqn. (32), \( \lambda \{ C_2 \} \) can be calculated using Eqn. (31) and the vector \( \lambda C \) is restated as

\[
\lambda C = \begin{bmatrix} 0 \\ \{ K_{21} \}[U_1] + [K_{22}][U_2] \end{bmatrix}
\]  

The solution of the problem may then be expressed symbolically as

\[
U_n = \lambda K^{-1} C = \frac{K^{-1} C}{C^T K^{-1} C}
\]

\[
\bar{q}_n = U^T K U = \left( \frac{K^{-1} C}{C^T K^{-1} C} \right)^T K \left( \frac{K^{-1} C}{C^T K^{-1} C} \right) = \lambda_n
\]

for the \( n \)-th iteration step. The iteration results from Eqn. (35) and (36) are used in a feedback loop to update \( K \) and \( \lambda \). The iteration will be terminated when the following convergence criterion is satisfied

\[
\frac{\| U_n - U_{n-1} \|}{\| U_n \|} \leq \delta_1
\]

\[
\frac{\| \bar{q}_n - \bar{q}_{n-1} \|}{\| \bar{q}_n \|} \leq \delta_2
\]

where \( \delta_1 \) and \( \delta_2 \) represent the desired accuracy of the solution.

COLLAPSE ANALYSIS OF A S-SHAPED FRAME
The limit analysis theory discussed above has been coded into a finite element program. This program is applied to the collapse analysis of a S-shaped frame. The results are compared to that obtained from an ABAQUS elasto-plastic analysis.

The shape and dimension of the S-shaped frame is shown in Fig. 1. One end of the frame is fixed, and the other end is subjected to a displacement boundary condition and limited to move only in the axial direction. The thickness of the frame is 0.7mm, and the material properties are

\[ \sigma = 200(1 + 83\varepsilon)^{0.21} \text{ (MPa)} \]

- Young’s modulus : 210 GPa
- Poisson ratio : 0.3

Only one half of the frame is analyzed due to its symmetry. The deformed shapes obtained from the limit analysis are compared with those from ABAQUS analysis as shown in Fig. 2. The results from both analyses are almost the same, and two plastic hinges occur in each case. The deformation is localized severely at those hinges. The load–displacement curve also shows a good agreement as shown in Fig. 3. The maximum collapse load is about 24.9kN in the elasto-plastic analysis, and 25.4kN in the limit analysis case as shown in Fig. 3. Table 1 lists the CPU time for each analysis. Considering the elasto-plastic analysis is run on a super computer, it is shown that the finite element limit analysis is a very effective way for structural collapse analyses.
The Audi space frame and Alcoa aluminum structure are studied in this work which are shown in Fig. 4. The front half of the structure is modeled as a simplified frame as shown in Fig. 5 and the initial base model is named as Model–I. The uniform square cross-section of 50mm x 50mm is used for the model. The initial yield stress of the material is defined as 200MPa, and the effective stress–strain relation is

\[ \sigma = 200(1 + 8\varepsilon)^{0.21} \text{ (MPa)} \]

A kinematic boundary condition is applied to the front end of the model in the direction to the rear, and the displacement is constrained only free in the straight back direction to simulate the real cases in a frontal car crash.

The deformed shape of Model–I is shown in Fig. 6(a) when the front end displacement is 200mm. Fig. 6 (b), (c), and (d) illustrate the velocity fields when the front end displacement is 100mm, 200mm, and 300mm, respectively. The direction and magnitude of the arrows in the figure represent that of the velocity at each node. It can be seen that the major deformed parts are the front side member, front pillar, and side sill. It should be noted that a large deflection could occur at the passenger compartment due to the severe deformation of the front pillar. The collapse load of Model–I is about 100kN as shown in Fig. 9.

In order to reduce the passenger compartment deflection and increase the collapse load, several design changes are proposed as listed in Table 2. The deformed shape and velocity field of Model–II are shown in Fig. 7 when the front end displacement is 200mm. The initial collapse load increases to 320kN as shown in Fig. 9. Although the load-carrying capacity increases remarkably, the deflection toward the passenger compartment is not reduced. This is because the front upper member hardly deforms, and it causes the passenger room in severe
To improve the crashworthiness of the structure, the front upper member is shorten by 100mm at its front. This Model–III is designed to reduce the deflection into the passenger compartment with the energy absorbed by the front side members. The deformation is concentrated at the front side members in Model–III as shown in Fig. 8, and the deflection toward the passenger compartment is reduced remarkably. The initial collapse load of Model–III is about 280kN, which is lower than that in Model–II.

It can be seen from Fig. 9 that the collapse load decreases gradually after the initial collapse since the structure becomes weaker and weaker as deformation proceeds. The initial collapse load and the slope in the load–displacement curve are important information. The area under the curve represents the energy absorbed during the deformation, from which the energy absorption efficiency can be evaluated. Figure 9 shows that Model–II has the largest initial collapse load and larger collapse load during the deformation than other models. On the other hand, Model–III has lower initial collapse load than Model–II does, but it has less distortion of the passenger compartment. These results indicate that there are some trade-offs in the vehicle structural design. Nevertheless, Model–III has the large load-carrying capacity and less distortion compared to other models, therefore, it should be considered as a good design in the sense of the quasi-static plastic collapse.

CONCLUSION

In this paper, collapse analysis is conducted using the finite element limit method. A S-shaped frame is chosen as an example to compare the limit analysis with the elasto-plastic analysis. Results show good agreements in the load-carrying capacity and deformation mode predictions. A simplified space-framed vehicle model is then studied for the optimum design process. Example studies show that the collapse load can be increased by reinforcing the weak parts of the structure. By changing geometry, structural deformations can be
relocated. The optimum design of a vehicle structure needs a large load-carrying capacity and low distortion of passenger compartments, which can be achieved by using the finite element limit analysis. This study demonstrated that the finite element limit analysis is a very effective tool for structural collapse analysis.

REFERENCE


Table 1. Comparison of the CPU time.

<table>
<thead>
<tr>
<th></th>
<th>ABAQUS Elasto plastic analysis</th>
<th>Limit analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>CPU time</td>
<td>2892 (sec)</td>
<td>9125 (sec)</td>
</tr>
<tr>
<td>Machine (Performance)</td>
<td>CRAY-YMP C90 (16GFLOPS)</td>
<td>HP C100 Workstation (48MFLOPS)</td>
</tr>
<tr>
<td>Estimated CPU time</td>
<td>964000 (sec) with HP C100 Workstation</td>
<td>9125 (sec) with HP C100 Workstation</td>
</tr>
</tbody>
</table>
Table 2. Design changes proposed.

<table>
<thead>
<tr>
<th>Model ID</th>
<th>Reference model for the change</th>
<th>Description and Design changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>–</td>
<td>Initial base model: simplified space-framed structure of Audi space frame and Alcoa aluminum structure</td>
</tr>
<tr>
<td>II</td>
<td>Model–I</td>
<td>The cross-section of the front pillar, side sill, and front side member are changed to 50mm x 100mm.</td>
</tr>
<tr>
<td>III</td>
<td>Model–II</td>
<td>The front upper member is shorten by 100mm at the front.</td>
</tr>
</tbody>
</table>
Figure 1. Dimension of the S-shaped frame studied.
Figure 2. Deformed shapes of the frame subjected to an end displacement.

(a) End displacement of 100 mm (Limit analysis)    (b) End displacement of 100 mm (ABAQUS)
(c) End displacement of 200 mm (Limit analysis)    (d) End displacement of 200 mm (ABAQUS)
Figure 3. Collapse load versus displacement.
Figure 4. Audi space frame and Alcoa aluminum structure.
Figure 5. Simplified Vehicle Model–I.

A - Front side member
B - Front upper member
C - Front pillar
D - Side sill
Figure 6. Deformed shape and velocity fields of Model–I under various front end displacements.
Figure 7. Deformed shape and velocity field of Model–II under the front end displacement of 200mm.
Figure 8. Deformed shape and velocity field of Model–III under the front end displacement of 200mm.
Figure 9. Collapse load versus displacement.