Optimum Shape Design of the Bus Window Pillar Member Based on B-Spline

Kim, Myoung Han
Student, Graduate School of Mechanical Engineering, Sungkyunkwan University
Suh, Myung Won, Bae, Dong Ho*
Professor, School of Mechanical Engineering, Sungkyunkwan University

The body structure of a bus is generally assembled by using various spot welded box members. Window pillar member is ordinarily built up with T-type member. This is a very important member connecting upper and lower structure of the bus body. However, it has been known that T-type member has some problems such as high stress concentration, low fatigue strength, and low structural rigidity. Because these problems cause degeneration of the performance and the durability of the bus body, there is need to improve them. In this study, a new approach for optimum design of the bus window pillar member is investigated. Stress distribution is estimated through finite element analysis. To optimize the shape of the outer gusset connecting the vertical and horizontal member of the T-type window pillar member, B-spline is applied. To verify the method, stress analysis on the spot welding points which are fatigue crack initiation points, is performed. It is found that the new approach could effectively alleviate stress concentration at the spot welding point and the outer gusset, and improve structural rigidity. Particularly, the new bus window pillar member designed through optimization has the following characteristics: 1) the configuration of the structure is very simple, and 2) fabrication process is simpler than the current model.

Key Words : Window Pillar Member, B-Spline Optimum Design, Stress Concentration, Spot Welding, Fatigue Strength, Structural Rigidity

1. Introduction

Recently, automotive manufacturers are getting more intersted in light weight vehicles. Especially, improvement of the body structure has become a big issue for fuel economy and reducing the environmental pollution from exhaust gas. However, fabrication of a light weight body is limited by constraints such as strength, stiffness and crash worthiness. There are many means toward light weight body such as design change, application of a new joining technology, and using lighter materials. Because the replacement of material or change of the joining technology may increase the manufacturing cost, reshaping of the body has been generally adopted. (Masahiro and Yukio, 1998)

However, simple reshaping of parts may decrease the structural rigidity and fatigue strength of the actual body structure as shown in Fig. 1 due to stress concentration around the spot welding points and the weakest discontinuity regions. Therefore, design and fabrication techniques to attain effective and optimized weight reduction while improving the strength and the structural rigidity are getting more important. On the one hand, the previous studies on shape optimization for reduction of vibration and noise

* Corresponding Author,
E-mail : bae@yurim.skku.ac.kr
TEL : +82-31-290-7443 ; **FAX** : +82-31-295-1937
Professor, School of Mechanical Engineering, Sungkyunkwan University 300 chunchun-dong, Jangan-ku, Suwon, Kyungki-do 440-746, Korea. (Manuscript **Received** April 27, 1999; **Revised** August 17, 2000)

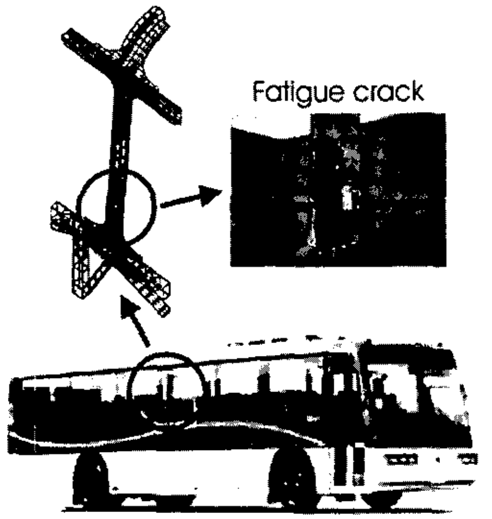


Fig. 1 The shape of a bus window pillar member and typical fatigue crack

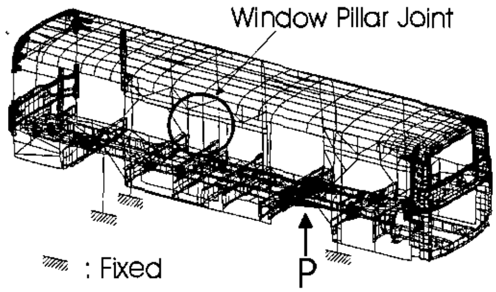


Fig. 2 Beam element model of the bus body

of the body and improvement of the fatigue strength and the structural rigidity used 3-dimensional beam elements when modeling body structure. Because the section property of beam elements are presented as results, formalization of the structure from the results requires more efforts in adopting actual design. (Hasegawa, 1992; Ereke and Yay, 1994; Kim, 1993)

T-type window pillar member in Fig. 1 is a very important structure to connect the upper and the lower parts of structure. However, it is the weakest region in the body structure of the bus. In this study, load pattern, deformation, and stress distribution of the window pillar members including the spot welding points are numerically studied. Using the results, a new optimum design technique for the window pillar member of the bus is proposed to improve its structural rigidity and fatigue strength.

2. Finite Element Analysis

2.1 Deformation of the bus body

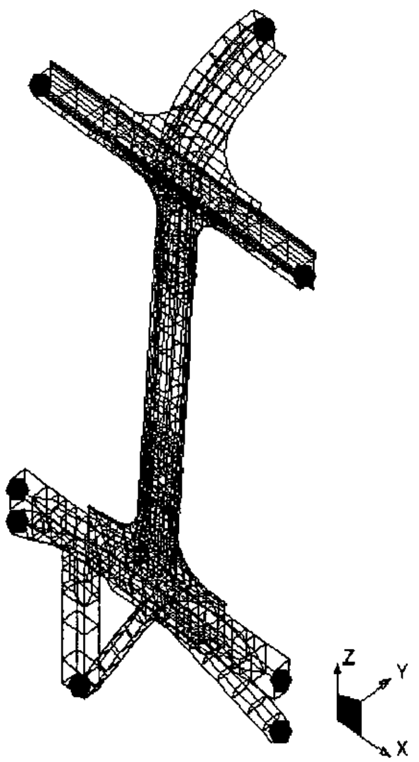
The main deformation modes of the bus body can be classified into bending and torsion. (Masahiro and Yukio, 1998) Usually, bending deformation is sufficiently restrained by a ladder type main frame. However, torsional deformation cannot be sufficiently prevented in case of the current typical bus structure due to wide and long inner space. The large torsional deformation directly influences the fatigue fracture of the window pillar members by the warping along the longitudinal direction of the body structure. An effective method against the torsional deformation of the bus body is to insert diaphragms which are perpendicular to the longitudinal direction. However, to maximize the inner space of the bus for passengers, there cannot be any diaphragm except the front and the rear wind shield glasses. Moreover, because the body structure of the bus must provide wider view to passengers, the window size has grown. (Lim and Shin, 1996) As the numbers and the cross section area of the window pillar member have decreased, the body structure has been put in more severe stress conditions.

2.2 Analysis model and conditions

Finite element analysis is performed to investigate the stress distribution of the window pillar member including the spot welding points. Firstly, the whole body structure was modeled with beam elements as illustrated in Fig. 2 to obtain the boundary conditions for the window pillar member shown in Fig. 3. Torsional load, which is known as the most dominant factor to the strength and stiffness of the window pillar member, is applied to the beam model. Two FAC's (front axle center) and one RAC (rear axle center) are fixed, and the load is applied to the other side of RAC to the vertical direction as shown in Fig. 2. From the results, loading conditions for the window pillar member shown in Fig. 3 are obtained. Four-node shell elements are used for the finite element analysis of the window pillar

Table 1 Mechanical properties of SPC

Tensile strength (MPa)	306.74
Yield strength (MPa)	164.64
Elongation(%)	58.5



● : Displaced points

Fig. 3 FEA model of the window pillar member subjected to the torsional loading of the bus structure

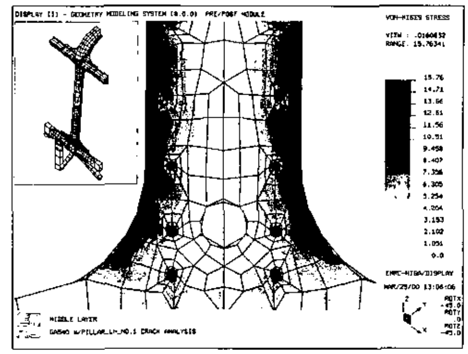


Fig. 4 Stress distribution subjected to torsional load

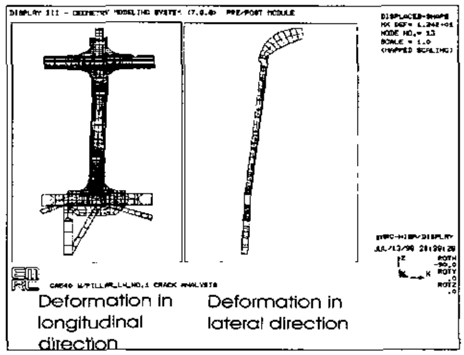
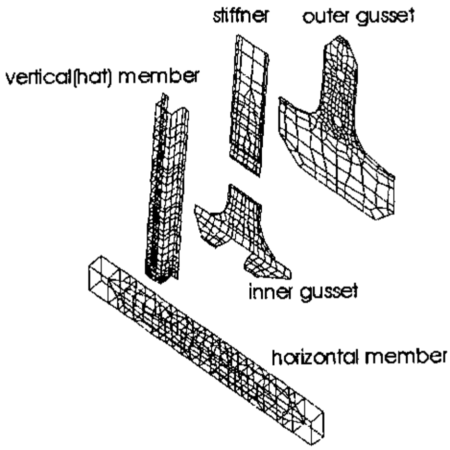


Fig. 5 Deformation subjected to torsional load

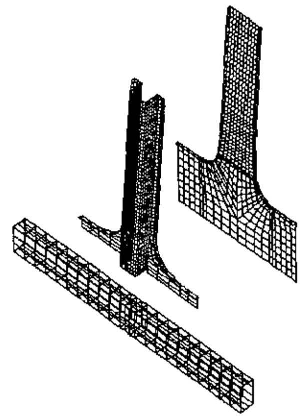
member. To analyze stress distribution around the spot welding points, 8-node solid elements are used, and finer meshes are generated to obtain accurate results. Spot welding nuggets whose diameter is 6 mm in structure are located with constant offset from outline of the outer gusset. The pitches of them are 70 mm on rectilinear part and 50 mm on fillet. The mechanical properties of the material shows in Table 1. FEM program used for the analysis is EMRC/NISA-II, Version 7.0(1997).

2.3 Result of analysis

Stress distribution and deformation of the window pillar member are shown in Fig. 4, and Fig. 5, respectively. Figure 6(a) illustrates the composition of the typical window pillar member. Its deformation to the longitudinal direction and the lateral direction are simultaneously revealed by warping as shown in Fig. 5. The maximum stress is generated at nugget edge of the spot welding point on the fillet of the outer gusset shown in Fig. 6, which is applied to restrain the stress concentration at the joint region between the vertical and the horizontal member. And, its position is coincided with the region where fatigue crack is generated as shown in Fig. 1. In this study, in order to estimate the influence of the longitudinal and the lateral displacement to stress concentration around nugget edge of the spot welding point, and to decide a mechanical main factor, for more effective analysis, stress analysis is performed on the two directional displacements, respectively.



(a) Initial design



(b) New model applying double arc

Fig. 6 Composition of the typical window pillar joint

From the results, it is shown that the maximum stress from the longitudinal displacement is larger by about four times than that from the lateral displacement, and its position is coincident with the actual crack. Therefore, the longitudinal deformation is chosen as the main mechanical factor that is concerned with fatigue failure from stress concentration generated at the nugget edge of the spot welded window pillar member. In this study, this longitudinal deformation is defined as an in-plane deformation.

3. The Shape Optimization of the Bus Window Pillar Member

As previously mentioned, the main mechanical factor which causes stress concentration is in-plane deformation or force. The maximum stress in the window pillar member from this stress is generated at the spot welding point around the fillet of the outer gusset. Because stress concentration and fatigue strength is considerably influenced by the shape of the outer gusset as shown in Table 2, and Fig. 7, we put the goal of this study in the shape optimization of the outer gusset. Since the composition of the joint member is very complicated, it is simplified as shown in Fig. 6(b) through the previous investigation. (Kim et al., 1997) The shape of the outer gusset of the current model has a single circular arc shown in Fig. 6 (a). By increasing the radius, the stress concentration at the spot welding point and the outer gusset

Table 2 The maximum stress according to the change of fillet radius

Fillet radius (mm)	Maximum stress on spot welding point (MPa)
80	151.2
82	143.7
84	136.8

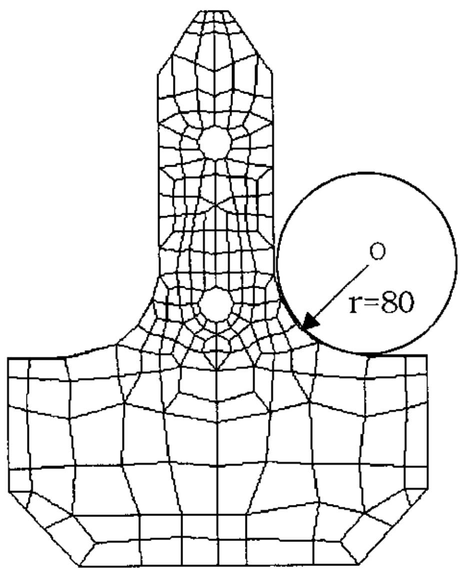


Fig. 7 Shape of outer gusset of the present model

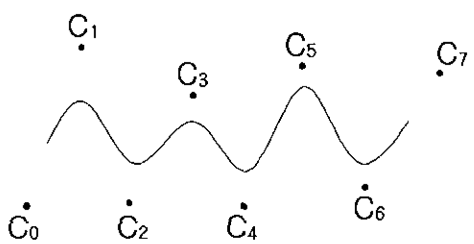
can be alleviated. However, increasing only its radius is not effective and resonable method in view point of its volume increase and obstruction to the passenger's view. In this study, a new shape of the outer gusset solving such problems is developed by using the optimum design technique.

3.1 The theory of optimization and its application

Applying an in-plane force on the window pillar member, the maximum stress occurs at the connecting area between the circular arc and the vertical tangential line. When this connection is smoothly transited by a proper control of the curve, it is expected that the stress concentration at the spot welding point and the outer gusset can be alleviated. (Masataka, 1967) Also, in this study, it is verified that the stress concentration at the spot welding point can be effectively alleviated as shown in Table 2. In order to modify the outline of the outer gusset, B-spline of the 3rd order which can freely govern the shape of the curve by changing the location of control points is used. B-spline is introduced from the following Eq. (1) (Anand, 1996)

$$P_i(t)=\frac{1}{6}[t^3\ t^2\ t\ 1]\begin{bmatrix}-1 & 3 & -3 & 1\\ 3 & -6 & 3 & 0\\ -3 & 0 & 3 & 0\\ 1 & 4 & 1 & 0\end{bmatrix}\begin{bmatrix}C_{i-1}\\ C_i\\ C_{i+1}\\ C_{i+2}\end{bmatrix},$$

$$0\leq t\leq 1,\ i=1,\ 2,\ \cdots n \tag{1}$$



Segment	control points
1	$C_0C_1C_2C_3$
2	$C_1C_2C_3C_4$
3	$C_2C_3C_4C_5$
4	$C_3C_4C_5C_6$
5	$C_4C_5C_6C_7$

Fig. 8 Control points of B-spline

where i is the sequence of segments

The curve made by n control points has $n-3$ segments. The segments are constructed by the combination of four consecutive control points as shown in Fig. 8. Even if the locations of a control point are abruptly moved, the nature of the algorithm keeps the curve smooth. As the B-spline is a versatile curve which not only sensitively reacts to infinitesimal change of location of the control points but also does not allow a mathematical discontinuity, it is able to represent freely the shape, and to be applied easily to manufacturing using the CNC machines. (Anand, 1996)

The optimization technique is applied to the half model of the window pillar member as shown in Fig. 9.

The problem statement is expressed as follows. (Arora, 1994; Rodriguez, 1997)

$$\underset{\vec{x}}{Min}:\ \underset{j=1,\cdots,l}{Max}\{\sigma_j(\vec{x})\} \tag{2}$$

subject to :

$$G(\vec{x})=Volume(\vec{x})-C\leq 0$$
$$x_i^l<x_i<x_i^u\ (i=1,\cdots,12)$$

The $\vec{x}$ is the design variable vector and σ_j presents the stress at the jth node on the outline. The components of the design variable $\vec{x}$ are x, y

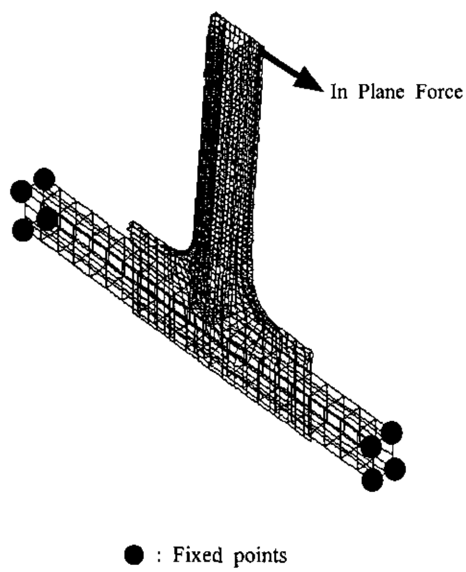


Fig. 9 FEA model of window pillar member for shape optimization

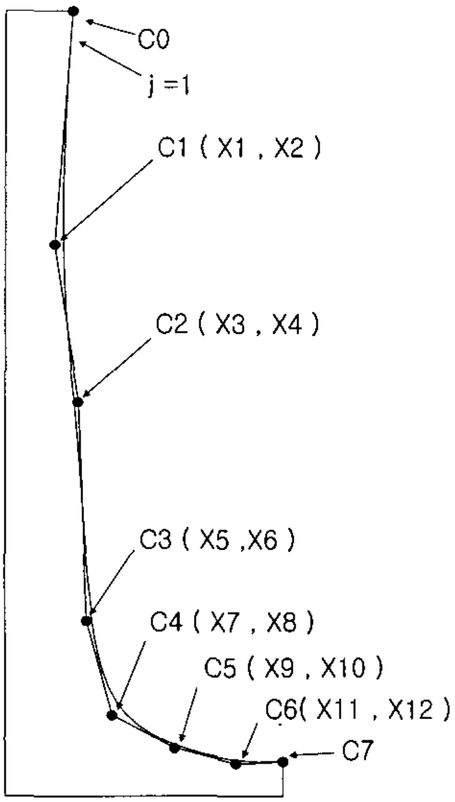


Fig. 10 Design variables and control points for the shape optimization

coordinates of the control points ($C_1 \sim C_6$) which determines the shape of B-spline as illustrated in Fig. 10. In this study, stresses on the 26 nodes, i. e., $l=26$, along the outline of the outer gusset are adopted. The volume of resource is constrained to be less than C which is taken as the weight of the current design of Fig. 6(a). The side constraints are set up to avoid impractical shape.

To solve Eq. (2) numerically, following transformation is conducted.

$$\underset{\vec{x}}{Min}\ \beta$$

subject to :

$$\begin{aligned} &g_i(\vec{x})=\sigma_j-\beta \le 0 \quad j=1,\cdots,l \\ &G(\vec{x})=Volume(\vec{x})-C\le 0 \\ &x_i^l<x_i<x_i^u \ \ (i=1,\cdots,12) \end{aligned} \tag{3}$$

The optimization procedure and algorithms are described in Fig. 11 and Table 3. (Arora, 1994; Rodriguez, 1997; Vanderplaats, 1985)

Table 3 The algorithms used in optimization

Whole routine algorithm	Augmented Lagrange Multiplier method
Search direction detecting algorithm	Broydon-Fletcher-Goldfarb-Shanno Variable metric method
One-Dimensional Search Algorithm	Golden Section method followed by polynomial interpolation

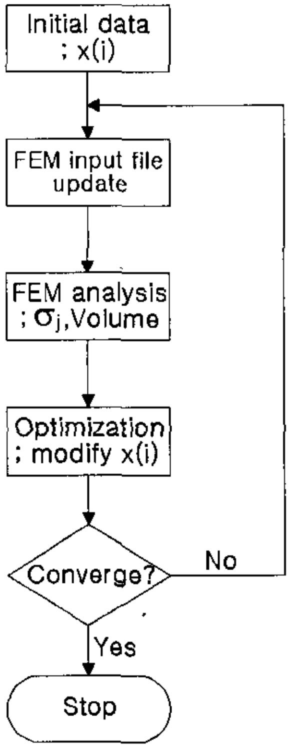


Fig. 11 Procedure of optimization

3.2 Result of the shape optimization

Stress distribution of the optimum model, and at the spot welding points on the vertical member are shown in Figs. 12, 13. The comparison of the results is shown in Table 4. It is shown that the maximum stress generated at the spot welding point in the hot spot area is considerably reduced by 43% in comparison with the initial model which is traditionally designed with a single circular arc. In addition, the maximum deformation as well as the rigidity of the member is also improved by 28%. This indicates that fatigue strength as well as stress concentration at the spot

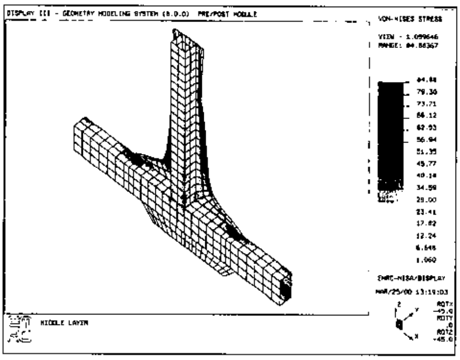


Fig. 12 Stress distribution and deformation of optimum model subject to in-plane load

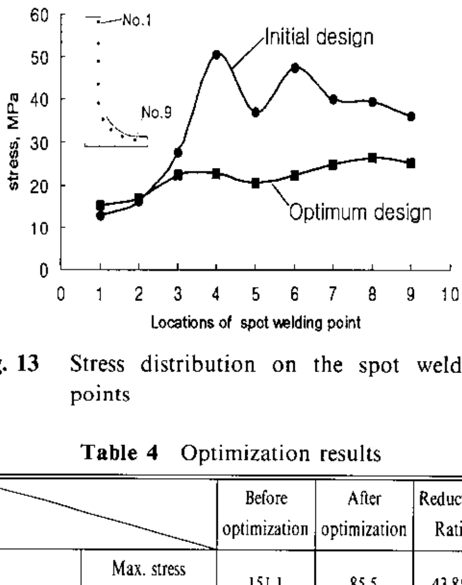


Fig. 13 Stress distribution on the spot welding points

Table 4 Optimization results

		Before optimization	After optimization	Reduction Ratio
In-plane Load (2940N)	Max. stress (MPa)	151.1	85.5	43.8%
	Max. deformation (mm)	0.52	0.37	28.8%
	Weight(kg)	16.5	16.48	0.15%

welding points and the outer gusset are considerably improved by the shape optimization. The shape of the optimum model compared to the initial one is shown in Fig. 14.

From these results, it is expected that the proposed new model of the spot welded window pillar member can be effectively and practically used in a bus body.

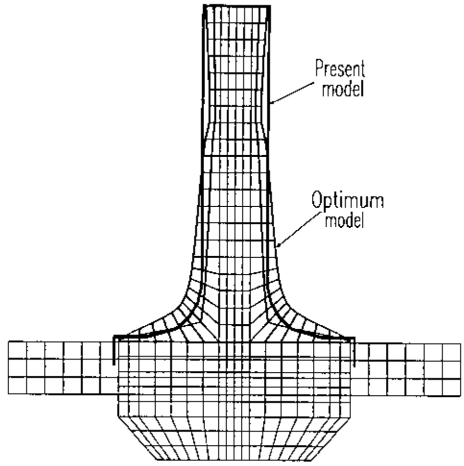


Fig. 14 The configuration of the optimum model

4. Conclusion

T-type window pillar is a very important structural member connecting the upper and lower structure of a bus body. However, it is the weakest region in the body structure of a bus. In order to develop a new optimum design for the bus window pillar member, load pattern, deformation, and stress distribution on the current window pillar member including the spot welding points are numerically studied. And then, to describe the shape of the gusset connecting the vertical and the horizontal members of the T-type window pillar member, the B-spline is used. this curve is optimized to attain simultaneously both alleviation of stress concentration at the hot spot regions and satisfaction of its structural constraint conditions. Obtained results are as follows;

(1) by changing the shape of the outer gusset, the maximum stress and the deformation decreased by 43%, and by 28%, respectively, under the same weight.

(2) it is confirmed that the optimization using B-spline is a very useful technique for shape optimization. The proposed approach is expected to be able to contribute to the improvement of fatigue strength and the structural rigidity of the bus body structure.

(3) particularly, the new bus window pillar member designed through optimization has the following characteristics; a) the configuration of

the structure is very simple, and, b) the fabrication process is not complicated in comparison with the current model.

Acknowledgement

The authors are grateful for the support provided by a grant from the Korea Science & Engineering Foundation(KOSEF) and Safety and Structural Integrity Research Center of the Sungkyunkwan University.

References

Masahiro, Shikida and Yukio, Kanayama, 1998, "The Strength of Automobile Body," AddTech, Korea, p. 93.

Hasegawa, A., 1992, "A Study of Vehicle Body Structure Using Optimization Analysis," *JSAE Review*, Vol. 52, No. 6, p. 988.

Ereke I. M. and Yay, K., 1994, "Computer Aided Design Analysis of Bus Structures," *Int. J. of Vehicle Design*, Vol. 5, No. 1/2, p. 67~85.

Kim, T., 1993, "Study on the Stiffness Improvement of Bus Structure," SAE 931995.

Lim, S. H. and Shin, D. M., 1996, "The Design of the Automobile," Hyungseol, Korea, p. 454.

Kim, M. H., Yim, S. H., Bae, D. H. and Tomioka, Noboru, 1997, "A Study on the Optimum Design of Bus Window Pillar Joint," *JSAE Proceeding No.* 974.

Masataka, Nishida, 1967, "Stress Concentration," Morikita, Tokyo, pp. 112~126.

Anand, Vera B., 1996, "*Computer Graphics and Geometric Modeling for Engineers*," John Wiley and Sons, Inc.

Arora, J. S., 1994, "*Introduction to optimum Design*," McGraw-Hill, Singapore, pp. 21~24.

Rodriguez, J., 1997. "Optimizing the Shape of Mechanical Elements and Structures," Marcel Dekker Inc., pp. 45~59.

Vanderplaats, G. N., 1985, "ADS-A Fortran Program for Automated Design Synthesis, Ver. 1. 10," Engineering Design Optimization Inc., Santa Barbara.