Springer

Journal of Mechanical Science and Technology

Journal of Mechanical Science and Technology 22 (2008) 1668~1676

www.springerlink.com/content/1738-494x
DOI 10.1007/s12206-008-0428-3

Topology optimization of compressor bracket

Jeong Woo Chang[1] and Young Shin Lee[2,*]

[1]*Halla Climate Control Corp., Daejeon, Korea*
[2]*Department of Mechanical Design Engineering, Chungnam National University, Director of BK21 Mechatronics Group, Daejeon, Korea*

(Manuscript Received July 3, 2007; Revised March 12, 2008; Accepted April 25, 2008)

Abstract

Topology optimization is very useful engineering technique especially at the concept design stage. It is common habit to design depending on the designer's experience at the early stage of product development. Structural analysis methodology of compressor bracket was verified on the static and dynamic loading condition with 2 bracket samples for the topology optimization base model. Topology optimization is able to produce reliable and satisfactory results with the verified structural model. Base bracket model for the topology optimization was modeled considering the interference with the adjacent vehicle parts. Objective function was to minimize combined compliance and the constraint was the first natural frequency over 250 Hz. Multiple load cases such as normal mode calculation and gravity load conditions with 3-axis direction were also applied for the optimization, expecting an even stress distribution and vibration durability performance. Commercial structural optimization code such as optistruct of Altair Engineering was used for the structural topology optimization. Optimization was converged after 14 iterations with the satisfaction of natural frequency constraint. New bracket shape was produced with the CATIA based on the topology optimization result. The new bracket from topology optimization result was compared with the traditional concept model and topology optimization base model under 4 load cases. 14 % 1'st natural frequency of new bracket with only 4 % mass increment increased compared to the concept model. 31 % mass decreased compared to the base model without the increment of stress under gravity load cases. It was analyzed thata new bracket would not fail during a vibration durability test, and these results were verified with a fabricated real sample under the durability condition.

Keywords: Topology optimization; Compressor bracket; Durability; Multi-objective optimization

1. Introduction

It is very important to reduce the development times from the initial concept development stage to the mass production stage in automotive engineering. Much trial and error occurs from the initial design to mass production to verify the performance and durability and other design criteria. Computational simulations for reducing such trials and errors are generally utilized and have proven to be useful tools in many areas. The ability of using CAD/CAE has become one of the core technologies nowadays because of shortened development periods.

The compressor mounting bracket was studied in this paper. The compressor plays a very important role in the automotive air conditioning system. It is attached to the engine via bracket and therefore the compressor bracket is exposed to the heaviest vibration conditions among the air conditioning parts. Because of heavy vibrating load conditions and the need for reducing weight to improve the fuel efficiency, the compressor bracket is one of the most important parts that requires optimum structural design.

The reliability of structural analysis results should be proved first of all for the optimum design iteration. Material properties, modeling methodologies and boundary conditions should be checked for structural

*Corresponding author. Tel.: +82 42 821 6644, Fax.: + 82 42 821 8844
E-mail address: leeys@cnu.ac.kr

analysis reliability and these efforts for the optimum design can give great effects at the concept or initial design stage of parts.

2. Homogenization method

The basic idea of structural topology optimization as shown in Fig. 1 is that a structural domain consists of many rectangular perforate materials, and these microstructures within design domain material are reproduced to maximize structural stiffness. As a consequence, the material distribution function is identified. In other words, topology optimization is to find the optimal material distribution function satisfying constrained condition in the whole design domain [1-4]. So, the objective of topology optimization is the strain energy of the design structure and the constraint function is the density of the material. Many types of combined structural loads could be considered to get the optimal structural shape satisfying objectives and constraints with the help of the development of many topology optimization algorithms [5-7]. Evolutionary optimization scheme [8-10] is another methodology under study to get the optimized structure configurations.

Ω_m is the material distribution domain and initial constraint condition, also. These relationships can be expressed as follows.

$$\text{Minimize } U = \frac{1}{2} E_{ijkl}(x)\varepsilon_{ij}\varepsilon_{kl}$$
$$\text{Subject to } Vol = measure\left(\Omega_m\right) \tag{1}$$

Generally, topology optimization begins with homogeneous distributed materials, and these materials are changed to get the different material densities. $E_{ijkl}(x)$ can be defined as follows.

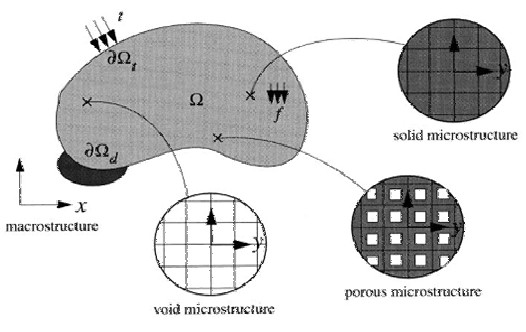

Fig. 1. Design domain and microstructure [6].

$$E_{ijkl}(x) = X\left(x\right)\overline{E}_{ijkl}(x) \tag{2}$$

$\overline{E}_{ijkl}(x)$ is the elastic modulus of material and $X\left(x\right)$ is the indicated factor which shows the presence of the materials.

But from Eq. (2) the presence of material would be distributed as discontinuous. So the elastic tensor is modified as Eq. (3) with the adoption of density function to get a smoothed material distribution within the design domain.

$$E_{ijkl}(x) = \xi\left(x\right)\overline{E}_{ijkl}(x) \tag{3}$$

Here $0 \le \xi(x) \le 1$ and $x \in \Omega$.

The volume of the material is as follows.

$$V_s = \xi\left(x\right)d\Omega \tag{4}$$

The density function can be expressed as $\xi(x) = 1 - a\left(x\right)b\left(x\right)$ in Fig. 2. and the size of empty space has the limitation of $0 \le a(x) \le 1$, $0 \le b(x) \le 1$.

If we assume the micro-structure as isotropic, matrix [D] can be expressed as the function of a and b as follows.

$$D = D^h\left(a,b\right) \tag{5}$$

Elastic modulus matrix considering the size of inner element $D = D^h\left(a,b\right)$ can be defined as follows.

$$D^h\left(a,b\right) = \left(1-ab\right)D = \rho\left(a,b\right)D \tag{6}$$

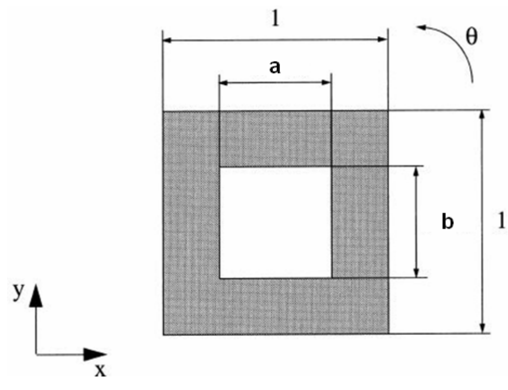

Fig. 2. Unit cell of 2D micro-structure [6].

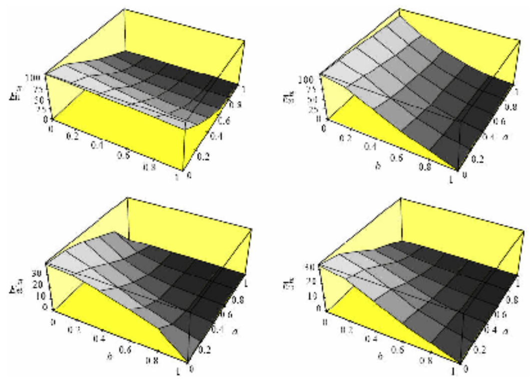

Fig. 3. A homogenized material constant matrix [7].

[D] matrix can be as follows.

$$D = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \tag{7}$$

θ becomes the design variable in case of micro-structure being not isotropic, and D^h can be as follows.

$$D^h = D(a,b,\theta) = R^T(\theta) D^h(a,b) R(\theta) \tag{8}$$

It is impossible to calculate with the finite element method after getting the elastic modulus $E_{ijkl}(x)$ of every design variable. As a consequence, it is calculated with Legendre-polynomial after calculating the homogenized elastic modulus $E_{ijkl}(x)$ of 36 cases (21 cases considering symmetric condition in reality) from the design variables a, b between (0, 1). Fig. 3 shows the homogenized material constant matrix.

3. Compressor bracket model verification

The verification of the structural model should be performed first of all to perform the structural optimization efficiently. So, the reliability of the structural model for finite element calculations was verified by the validation of experiments and calculation results under static and dynamic load conditions. Compressor assembly model is composed of compressor, bracket and bolts as shown in Figs. 4, 5. Engine mounting areas of bracket are constrained with all DOFs.

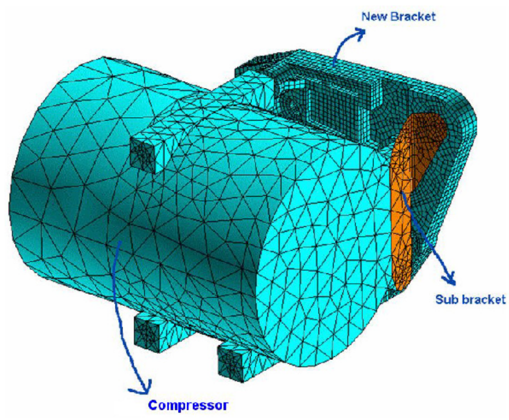

Fig. 4. Structural analysis model of bracket assembly.

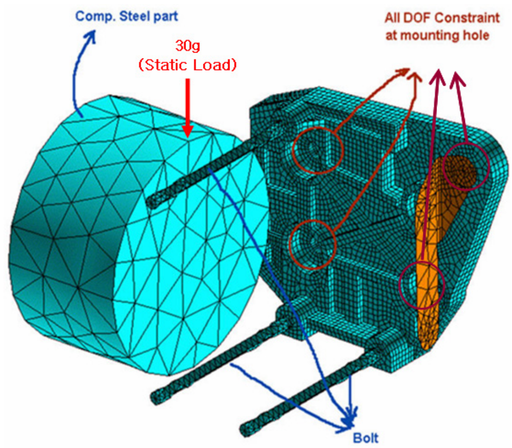

Fig. 5. Boundary conditions of bracket assembly.

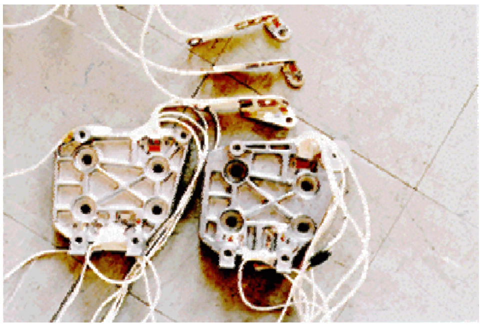

Fig. 6. Two bracket samples with strain gage attachment.

3.1 Static load case

Two brackets were prepared to get the validation between experiment and analysis under static load condition. Eight strain gages were attached to each bracket as shown in Fig. 6 and static load was applied to the compressor part as shown in Fig. 7. Structural analysis was done with the same static load condition, and the stress calculation results are shown in Figs. 8, 9.

The results show that the modeling method of the compressor assembly is quite reasonable and reliable. Commercial structural analysis code (ANSYS) was used for these analyses. The material properties of bracket as shown in Table 1 were ADC10, which is usually used in aluminum die casting material.

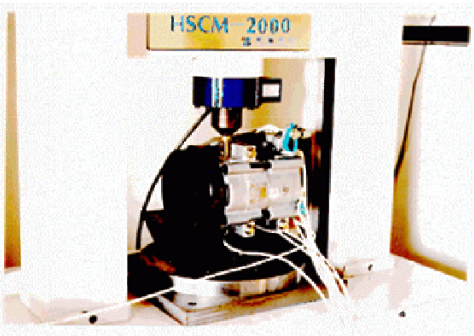

Fig. 7. Static load experiment configuration of bracket assembly.

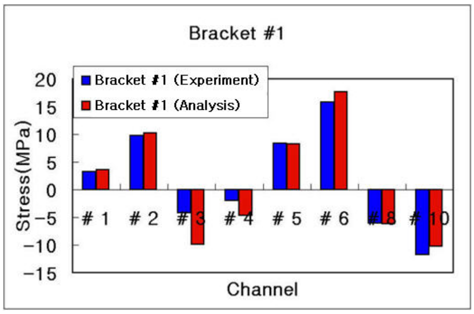

Fig. 8. Static load experiment results of bracket #1.

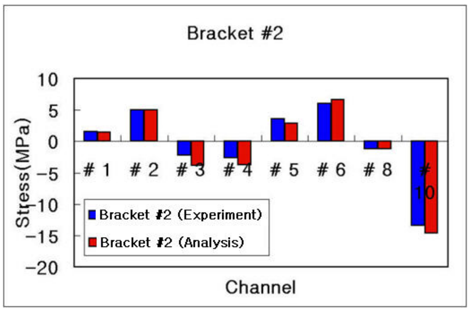

Fig. 9. Static load experiment results of bracket #2.

3.2 Dynamic cases

The compressor is attached to an automotive engine via bracket and is exposed to severe vibrations. So, the reliability of the vibration characteristics is especially important for the compressor assembly. Two vibration characteristics of bracket component and compressor assembly were verified to get the reliability of structural analysis results.

3.2.1 Bracket component

Modal experiments of single bracket were performed with free-free condition and the similar structural normal mode analyses were done. Natural frequencies are shown in Table 2. The mode shapes as shown in Figs. 10-13 were compared to confirm the reliability of structural finite element modeling method. The first mode shape of the bracket was quite similar with the twist mode of the plate, and the second bracket mode shape was quite similar with the plate bending mode. The results show that the modeling of the bracket is well defined to calculate the vibration characteristics, and the results are quite reliable.

Table 1. Material properties of bracket (ADC10) [12].

	ADC 10
Elastic Modulus	74,000 MPa
Poisson's ratio	0.33
Density	2700 Kg/m^3
Yielding Stress	186 MPa
Tensile Strength	331 MPa
Fatigue Limit	131 MPa

Table 2. Modal experiment and analysis results of brackets with free-free condition.

(Hz)

Natual frequency	Bracket #1		Bracket #2	
	Test	Analysis	Test	Analysis
1'st	1396	1503	1912	1941
2 nd	1944	2094	2760	2829
3 rd	2488	2704	3328	3467
4'th	3280	3730	4784	5078
5'th	3848	4182	5800	6042

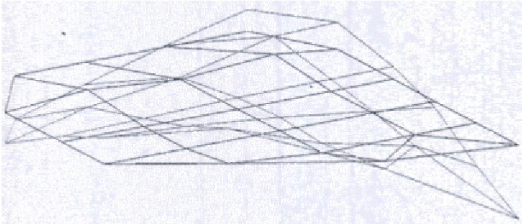

Fig. 10. Experimental first mode shape of bracket #2.

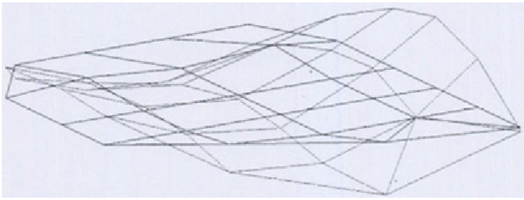

Fig. 11. Experimental second mode shape of bracket #2.

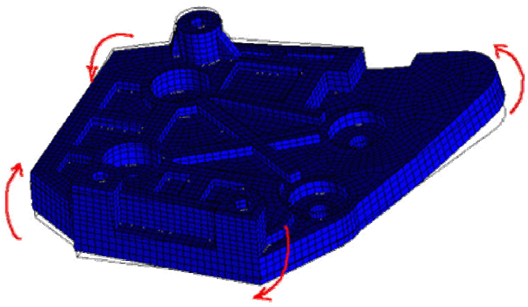

Fig. 12. Analytical first mode shape of bracket #2.

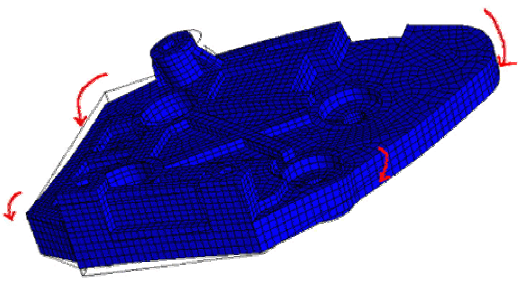

Fig. 13. Analytical second mode shape of bracket #2.

3.2.2 Compressor assembly

The verification of compressor assembly vibration characteristics was performed by using vibration sweep with the bracket assembly fixed with fixture on the large shaker as shown in Fig. 14. The acceleration data of the compressor during the frequency sweep of

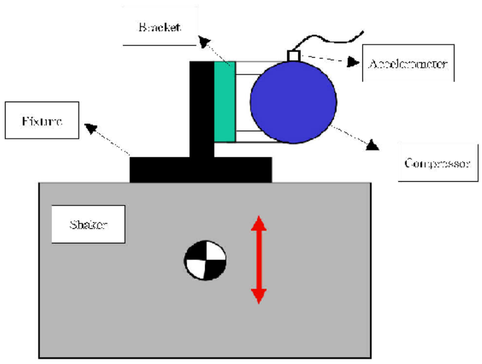

Fig. 14. Vibration durability experiment set-up of compressor bracket assembly

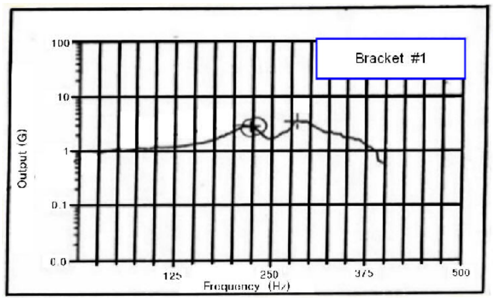

Fig. 15. Harmonic excitation experimental result of bracket #1 assembly.

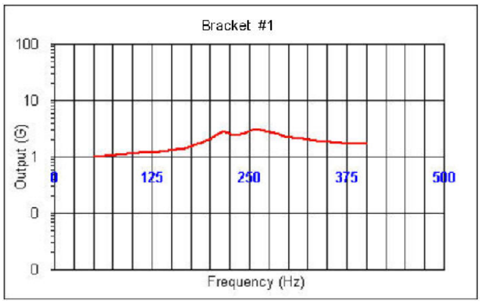

Fig. 16. Harmonic excitation analytical result of bracket #1 assembly.

vibration is compared with experimental results and analytical results under same conditions as shown in Table 3 and Figs. 15-18.

The results show that the modeling of the bracket assembly is well defined to calculate the vibration characteristics, and the results are quite reliable. Commercial structural analysis code (ANSYS) was used for these vibration analyses.

Table 3. Harmonic excitation experimental and analytical results of bracket #1 and bracket #2 assembly.

Natual frequency	Bracket #1		Bracket #2	
	Experiment	Analysis	Experiment	Analysis
1'st	222	216	239	245
Output G	2.84	2.74	2	2.05
2'nd	282	255	342	343
Output G	3.46	3	4.02	4.03

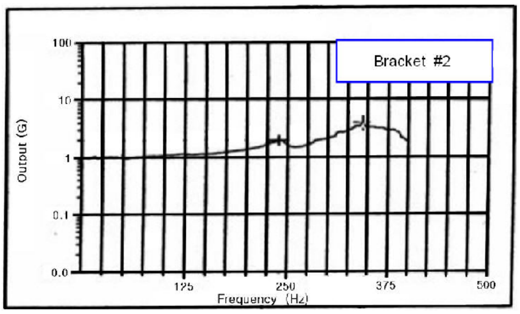

Fig. 17. Harmonic excitation experimental result of bracket #2 assembly.

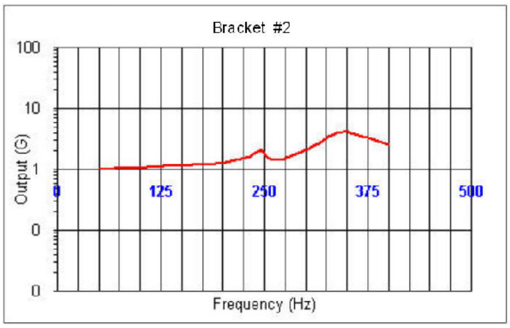

Fig. 18. Harmonic excitation analytical result of bracket #2 assembly.

4. Compressor bracket optimization

Generally, the bracket mounting positions would be determined without the interferences with other adjacent components. The initial concept design of a bracket is briefly determined by considering these mounting positions. It was a common habit of adding ribs among these mounting points without considering stiffness and durability of bracket and making iterative trial and error efforts based on the test results at the initial automotive development stage. It is

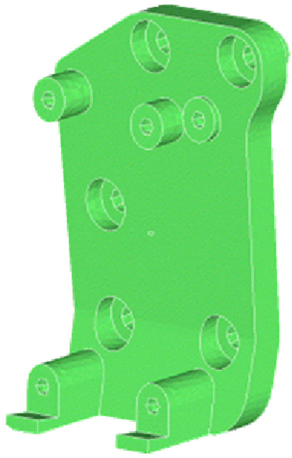

Fig. 19. Compressor bracket model of initial concept stage.

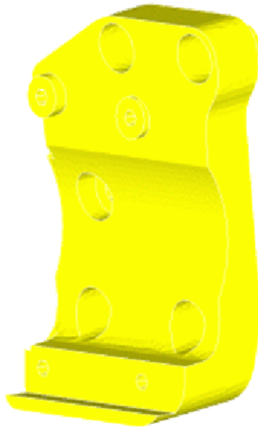

Fig. 20. Compressor bracket model for topology optimization

strongly dependent upon the experiences and insights of the development engineer to these iterative efforts.

It was possible to minimize these iterative efforts and to get a valid ground for the initial design with the help of a structural optimization method such as the topology optimization method. It is usual to get the maximized volume of structure without the interferences for the initial modeling of topology optimization. Typical initial concept design of compressor bracket was shown in Fig. 19 and base model of compressor bracket for topology optimization based on this concept design is shown in Fig. 20.

The boundary conditions for the optimization are the same with the compressor assembly analytical model as shown in Fig. 5.

There are several important things to consider when we are doing the structural topology optimization.

(1) Mesh Densities: Topology optimization results like general finite element analysis can be affected by the mesh densities of structure. So it has adequate mesh densities considering the structure feature which is being analyzed.

(2) Checker-board effects: Sometimes it would be hard to get the specific optimized shape after topology optimization. So the feature of the optimization code is needed in order to overcome these phenomena [11].

(3) Minimum member size: Topology optimization could give very small thickness of structure, which is hard to produce. Thus, the feasibility of manufacturing and effectiveness of the analysis at the same time needs to be considered.

(4) Manufacturing feasibility: Manufacturing methodology of structure according to the structure feature is usually pre-determined such as extrusion, casting. Topology optimization should give the results which are possible to enable manufacturing.

(5) CAD export capability: The results of topology optimization should be applied to the CAD program for manufacturing in many cases. It could be one of the important features for topology optimization tool.

Generally, the mass of the compressor bracket at initial concept stage is not important and the stiffness effect is not considered at all. The first role of the bracket is the stiffness, though the mass is important also. Mass reduction efforts are usually tried after getting enough stiffness during the development of automobile parts.

The optimization problem was formulated as follows: minimizing the combined strain energies of both dynamic conditions (1'st natural frequency) and the static 30 g in X, Y, Z directions was defined as objective function and the constraint function was defined to get the 1'st natural frequency over 250 Hz under fixed condition similar to the mounting condition of a bracket to a real vehicle. This problem is defined as containing multi-objective characteristics considering the static and dynamic conditions at the same time. 250 Hz as constraint represents the resultant frequency of 7500 rpm on the general 4-cylinder vehicle. 30 g as the other constraint represents the generally evaluated vibrating force coming from the vehicle engine. Commercial structural optimization code (optistruct) of Altair Engineering was used to solve the optimization problem.

5. Observations

A converged solution of topology optimization was achieved after 14 iterations. Iterative history of the optimization is shown in Figs. 21, 22. The resultant shape of the bracket is shown in Fig. 23. The new shape of the bracket, which is the result of the structural topology optimization, was made in CATIA as shown in Fig. 24. Structural analyses with concept bracket, base bracket and optimized bracket were performed to see the usability of topology optimization as shown in Table 4.

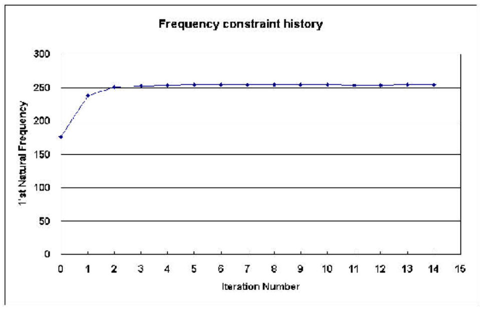

Fig. 21. Frequency constraint history of bracket topology optimization.

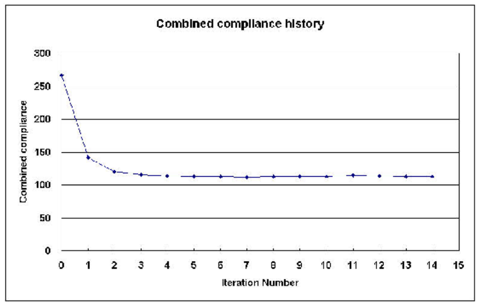

Fig. 22. Combined compliance history of bracket topology optimization.

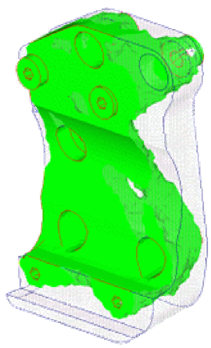

Fig. 23. Topology optimization result of bracket.

Table 4. Bracket model analytical results summary.

	Concept	Base	Optimized
Mass(g)	906	1366	941
1'st Natural Frequency(Hz)	222	256	251
30G Gravity - X(MPa)	18.6	11.4	14
30G Gravity - Y(MPa)	21.7	14.4	14.8
30G Gravity - Z(MPa)	25.6	14.4	11.5

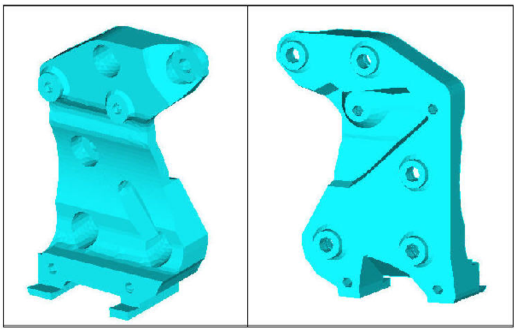

Fig. 24. Re-generated bracket shape from topology optimization result.

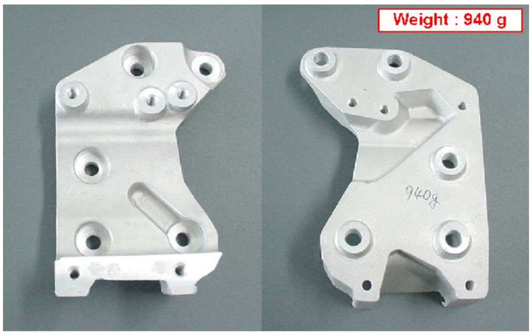

Fig. 25. Fabricated bracket sample based on topology optimization result.

Optimized bracket was fabricated based on this result as shown in Fig. 25, and a vibration durability test was performed with this optimized bracket. The bracket shows very stable vibration characteristics considering the output acceleration as shown in Fig. 2, and there was no structural failure after the durability test.

6. Conclusions

Design capabilities at the initial concept design stage play essentially important roles in the automotive parts design. The wrong design direction leads to very expensive costs and long times to recover to the right design direction. As a result, researches and developments for the design capabilities to get the initial robust design could affect the competitiveness of the products in these global and extremely competitive times.

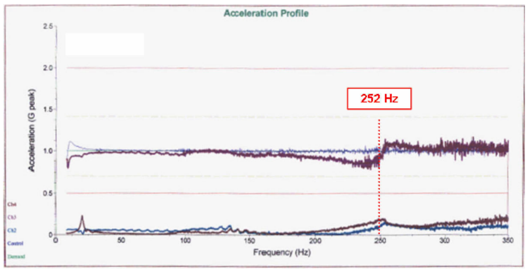

Fig. 26. Experimental natural frequency of fabricated bracket.

Topology optimization method of the structure can give the right design direction at the early development stage as studied in this paper. Those conclusions could be summarized with the topology optimization application to the compressor bracket.

(1) It was possible to propose an initial robust bracket design without interference to the adjacent parts, and the robustness of the proposed bracket, which doesn't show partial stress concentrations, was confirmed by succeeding structural analysis and the vibration durability test.

(2) It was confirmed that the optimized bracket with a 35 g mass increment compared to the concept design bracket increases 1'st natural frequency to 29 Hz above and decreases the averaged stress to 40% lower under the static load through the succeeding structural analysis. Thus, it shows that the topology optimization approach gives a very effective initial design with the minimized mass increment.

(3) It could be concluded that the multi-objective approach considering both static and dynamic condition gave a robust initial design and satisfactory vibration durability test result.

(4) The detailed verification of structural analysis model and reasonably defined optimization formulation would effect a reliable and satisfactory optimization result.

References

[1] M. P., Bendsoe and N. Kikuchi, Generating optimal topologies for structural design using a homogeniza-tion method, *Computer Methods in Applied Mechanics and Engineering*, 71 (1988) 197-224.

[2] B. Hassani and E. Hinton, A review of homogenization and topology optimization I – Homogenization theory for media with periodic structure, *Computers and Structures*, 69 (1998) 707-717.

[3] B. Hassani and E. Hinton, A review of homogenization and topology optimization II – Analytical and numerical solution of homogenization equations, *Computers and Structures*, 69 (1998) 719-738.

[4] B. Hassani and E. Hinton, A review of homogenization and topology optimization III – Topology optimization using optimality criteria, *Computers and Structures*, 69 (1998) 739-756.

[5] Z. D. Ma, N. Kikuchi and H. C. Cheng, Topological design for vibrating structures, *Computer Methods in Applied Mechanics and Engineering*, 121 (1995) 259-280.

[6] S. J. Min, S. Nishiwaki and N. Kikuchi, Unified topology design of static and vibrating structures using multiobjective optimization, *Computers and Structures*, 75 (2000) 93-116.

[7] J. H. Lee, Topology Design of a Structure with a Specified Eigen Frequency, Master's Thesis, Hanyang National University, Korea (2001).

[8] C. H. Ryu, A Study on the Ranked Bidirectional Evolutionary Structural Optimization Method for Fully Stressed Structure Design, Ph. D. Thesis, Chungnam National University, Korea (2002).

[9] Y. S. Lee and C. H. Ryu, A study on ranked bidirectional evolutionary structural optimization (R-BESO) method for fully stressed structure design based on displacement sensitivity, *Journal of Mechanical Science and Technology,* 21 (12) (2007) 1994-2004.

[10] H. S. Kim and Y. S. Lee, Optimization design for reduction of the sloshing using evolutionary method, *Journal of Mechanical Science and Technology,* 22 (1) (2008) 25-33.

[11] W. K. Kim, A Study on the Topology Optimization of Structure by using Digital Image Process Filter, Master's Thesis, Chungnam National University, Korea (2000).

[12] Sambo, The Characteristics of Aluminum Diecasting, Sambo Industrial Co. Ltd, Korea (1995).