Contemporary Engineering Sciences, Vol. 9, 2016, no. 20, 981 - 988
HIKARI Ltd, www.m-hikari.com
http://dx.doi.org/10.12988/ces.2016.66101

Safety Design of Automotive Seat Frame through Structural Analysis with Fatigue and Vibration

Jae-Ung Cho

Division of Mechanical & Automotive Engineering, Kongju National University
1223-24, Cheonan Daero, Seobuk-gu, Cheonan-si, Chungnam, 31080
Republic of Korea

Abstract

In this research, the safeness of automotive seats by structural analysis and a vibration analysis were studied. The pursuit of the automotive’s frame has is weight reduction. At the same time, in order to secure safeties of both the drivers and passengers, and structural and vibration analyses were conducted. Also, three models were made for designing the optimal automobile’s seat frame; basic model A, which is a seat frame that is actually used; model B, which doubled the thickness of connection rods at the hinge part; and model C, which doubled the thickness of the top side frame. In the structural analysis, for all three models the top frame showed a large amount of strains, and in the fatigue analysis, for all three models from the hinge parts of the top side of the frame showed the least durability. In the vibration analysis, for equivalent stresses at each model’s critical frequency, model C showed the more stable equivalent stress result by being compared with models A and B. When the three models were analyzed and compared, it could be shown that model C showed the more stable structure compared with the other models.

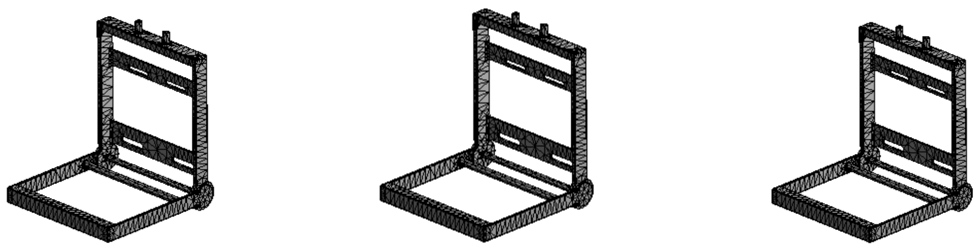
Keywords: Structural Analysis, Automotive Seat Frame, Fatigue and Vibration, Safety Design

1 Introduction

For all automotives, vibration reduction technology has been developed dramatically, and the seat with which the passenger has most contact is being evaluated and improved with various methods [1] [2] [3] [4] [5] [6] [7]. For this reason at designing vehicles, the complexities of their materials and manufacturing methods are increasing in order to attain the weight reduction of vehicle frames. At the same time, to safety secure of drivers and passengers, and designs through structural analysis and vibration analysis must be made precisely. Automotive seats play an important part in relation to passengers' safety and satisfaction. Seats must not only support the passengers, absorb vibrations from the road, and minimize the fatigue, but also have weight reductions and economical structures as much as possible. Generally, the seats take about 3 to 5% of weight and price of the total vehicle. Therefore, at the automotive industry, from the aspect of improving fuel efficiency and ensuring lasting competitiveness, much effort is being put into research that ensures not only the minimization of seats' price and weight but also comfortableness and stableness at the same time. It is modelled by using CATIA, referring to automotive seat frames that are currently in use. 3D-modeled seat frames are carried out with structural, fatigue, and vibration analyses by using ANSYS program. With these results, this study researched about stable designs by comparing A, which is the actual model, with B and C, which are virtually made models [8].

2 Study Result

The model's figure, referring to the actual automotive seat frame, was modeled in a 3D form in accordance with the actual size by using CATIA. The meshes of seat frame models are shown on Figure 1.



(a) (b) (c)

Fig 1. Mesh configuration of each model

Models A, B, and C have the same fixing conditions. Model A is equivalent to the actual automotive seat frames. Model B is equivalent to model A except for the fact that its supporting rods for the hinge part have double thickness. Model C is equivalent to model A except for the fact that its top frame's thickness is doubled. Model A is the only model based on the actual vehicle's seat frame. models B and C are models that are modified to suit to the optimal design. A static structural analysis was conducted for the models of this research. The structural analysis was conducted by using the boundary condition and the static loads. This analysis can derive the model's displacement, deformation, stress, and reaction. Also, by using a safety factor, the analysis can check its plasticity and assess whether it is destroyed or not. This research's models were analyzed by using ANSYS. Figure 1 shows the mesh of each model. For model A, it has 17,451 nodes; For model B, it has 17,682 nodes; For model C, it has 17,010 nodes. In case of elements, model A has 8,807, model B has 8,963, and model C has 8677. These models were simply analyzed by the general structural steel. It was assumed that the automotive seat frames were fixed at the lower part. This condition is the same with other models. Figure 2 describes the fixing condition of the seat frame.

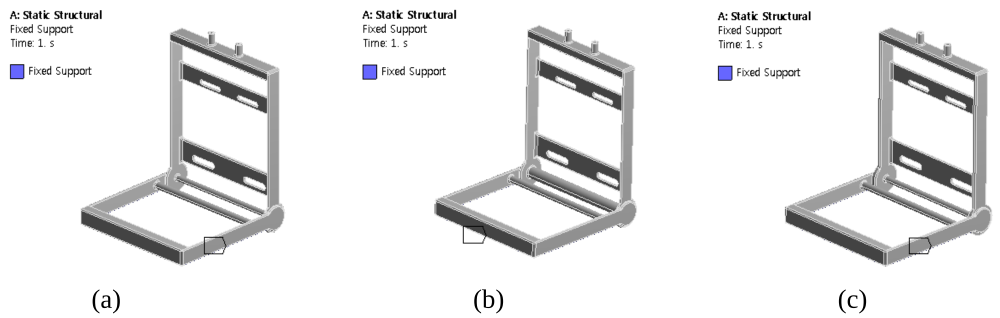


Fig 2. Model's fixed support: (a) model A, (b) model B, (c) model C

When a person seats on the automotive seat, the according force is applied to the seat's –Z axis. When a person leans on the seat, the according force will be applied to the seat's –X axis. Assuming that an average person weighs around 63Kg, the force that was applied to the seat was 620N towards –Z axis, and 250N towards –X axis. Figures 3, 4 and 5 describe models A, B and C, accordingly, with the forces applied to each model.

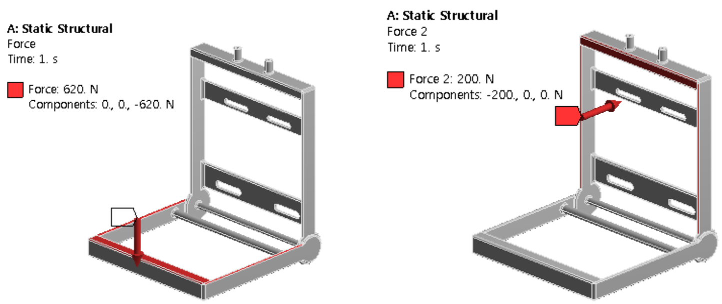


Fig 3. Model A applied with force

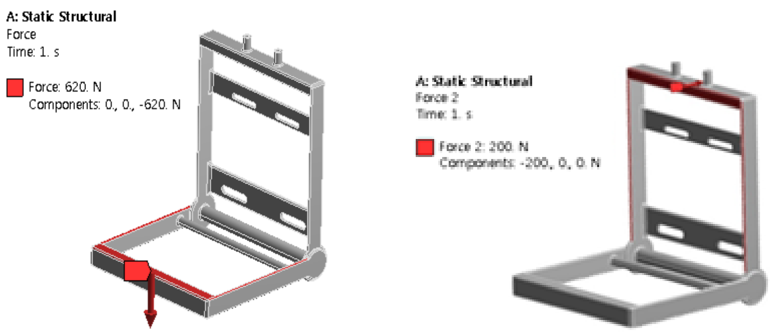


Fig 4. Model B applied with force

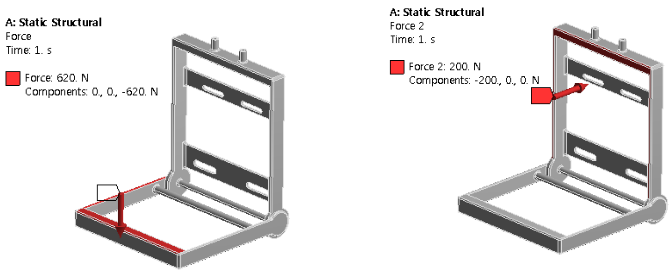
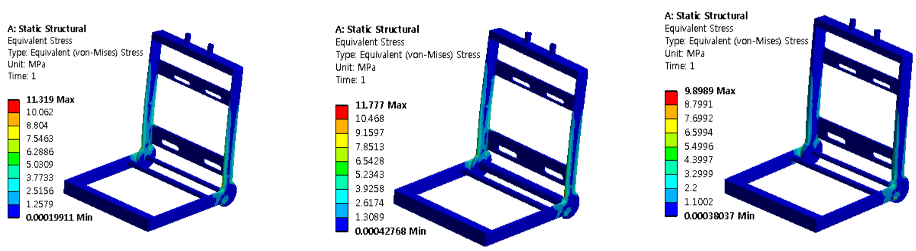


Fig 5. Model C applied with force

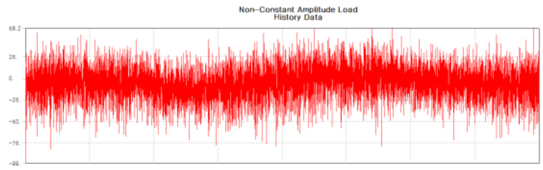
The equivalent stresses of models were compared with each other. As seen in Figure 6, for each model's equivalent stress, the maximum equivalent stress was 11.319MPa, 11.777MPa, and 9.8989MPa.



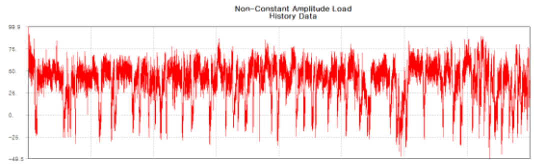
(a) (b) (c)

Fig 6. Results of equivalent stresses: (a) model A, (b) model B, (c) model C

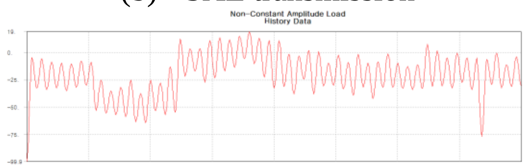
Structures with fatigue cycles of loads and displacement suffers cracks. These cracks will grow until the structure is finally destroyed. This phenomenon is called 'Fatigue failure'. To prevent fatigue damages of the structure against cyclic loads, a fatigue analysis was conducted by using ANSYS. For models A, B, and C, the same boundary condition as in the structural analysis was given. As seen at Figure 7, each model's life was calculated by using 'SAE bracket history', 'SAE transmission', and 'Sample history', which are nonuniform amplitude loads, and the life was compared as seen in Figures 8, 9 and 10.



(a) SAE bracket history

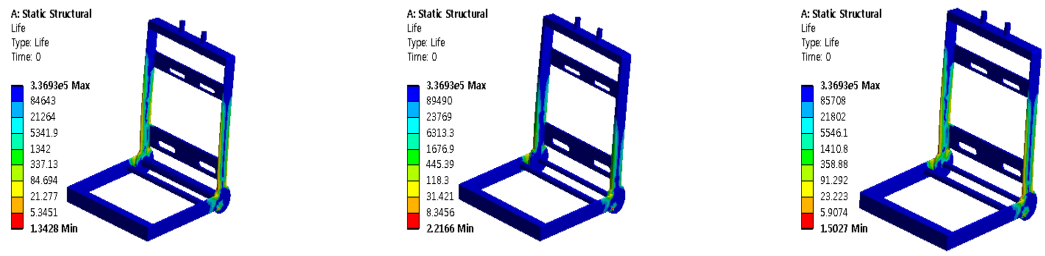


(b) SAE transmission



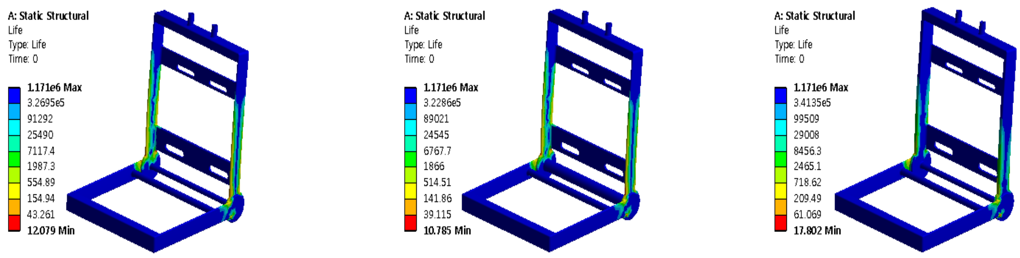
(c) Sample history

Fig 7. Nonuniform amplitude loads



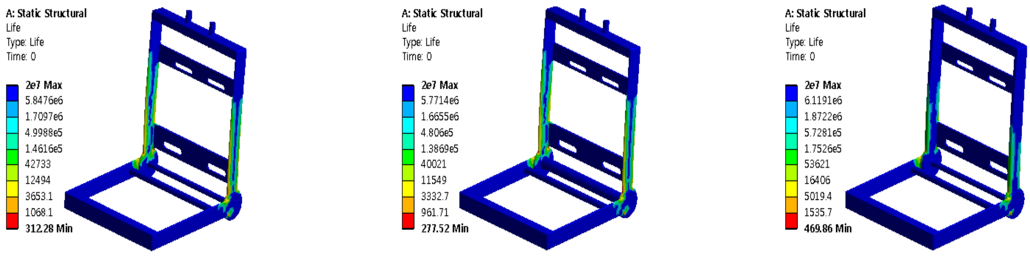
(a) (b) (c)

Fig 8. Life analysis results by using SAE bracket history: (a) model A, (b) model B, (c) model C



(b) (b) (c)

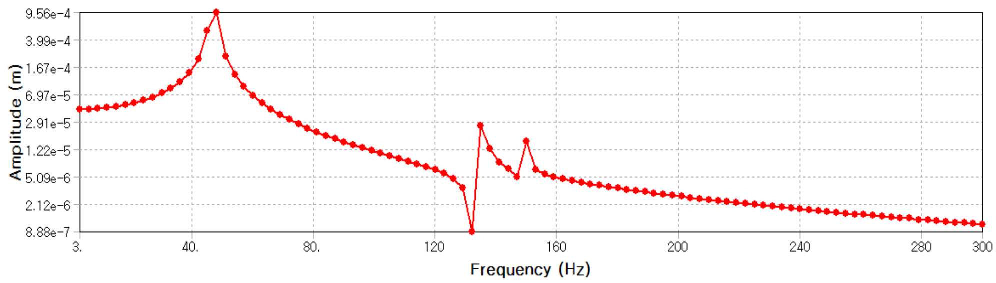
Fig 9. Life analysis results by using SAE transmission



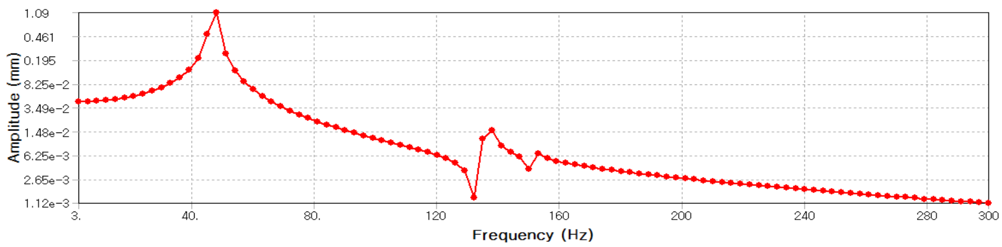
(a) (b) (c)

Fig 10. Life analysis results by using Sample History: (a) model A, (b) model B, (c) model C

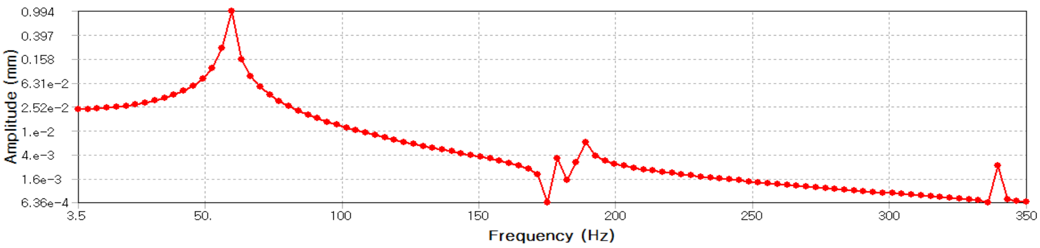
Even when different nonuniform amplitude loads were applied, the life analysis result showed that all three models had the fatigue damage from the hinge part up to the top frame. Models A and B, which were the models with different rod thickness of the hinge part, had no big differences. However, model C, the model with it's top frame having a double thickness, showed a result with the more stable fatigue life than the other two models. The same boundary conditions as in the structural analysis were used for the models in this research. A total of six total deformations were added to models A, B, and C by the modal analysis, and the mode was set from 1 to 6 at each total deformation item. The frequency ranges were obtained by modal analyses of three models and their frequency responses were obtained as shown by Figure 11. By obtaining the critical frequency from the harmonic response analysis, the equivalent stresses were obtained for each model at its critical frequency at Figure 12. The maximum values of equivalent stresses at critical frequencies were 250.3MPa for model A, 295.51MPa for model B, and 394.13MPa for model C. It could be seen that models B and C maximally act higher than model A.



(a) Model A

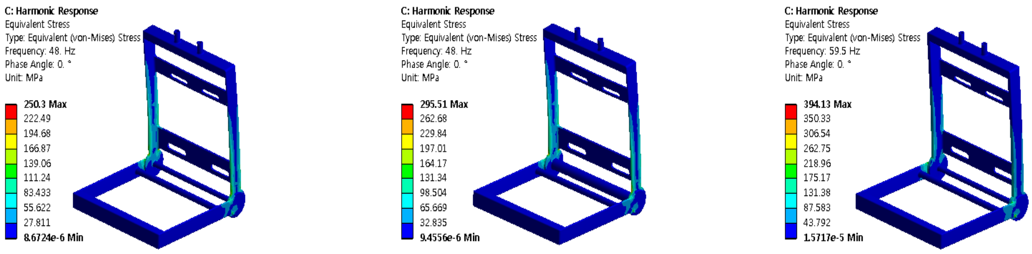


(b) Model B



(c) Model C

Fig 11. Frequency responses of models



(a) Model A (b) Model B (c) Model C

Fig 12. Equivalent stress analyses at each model's critical frequency

3 Conclusion

According to this research, for the optimal design of the automotive seat frame, by using actually used seat frames and CATIA, three models were made; model A, which is a basic model, model B, which is a model with double thickness connection rods, and model C, a model with a double thickness top frame. By using ANSYS program, structural and vibration analyses were conducted. The result of structural analysis showed that model C, compared with models A and B, showed less equivalent stress. The result of the fatigue analysis showed that all three models had the least amount of life at the part from the hinge part of the frame to the top frame, and all three models showed a stable state against fatigue damages. Model C had a more stable result compared with models A and B in relation to its fatigue life. The result of the vibration analysis showed that for the models' equivalent stresses at each model's critical frequency. Model C showed a more stable equivalent stress result than models A and B. After comparing the structural, fatigue, and vibration analyses of three models, model C showed a comparably stable structure than the other models.

Acknowledgments. "This work was (partly) supported by Advanced Motor Parts Regional Innovation Center (AMP.RIC) of Kongju National University administered by MKE (Ministry of Knowledge Economy), Korea."

References

[1] H. Hu, W.J. Lu, Z. Lu, Impact crash analyses of an off-road utility vehicle - Part II: Simulation of frontal pole, pole side, rear barrier and rollover impact crashes, *International Journal of Crashworthiness*, **17** (2012), no. 2, 163-172. http://dx.doi.org/10.1080/13588265.2011.632858

[2] H. Cho, J. Cho, K. Kim, D. Choi, Structurally safe design of rear seat frame applied with high tension steel plate, *International Journal of Digital Content Technology and its Applications*, **7** (2013), no. 12, 444-450.

[3] J. Zhan, M. Fard, J. Reza, A model to assess the comfort of automotive seat cushions, *International Journal of Occupational Safety and Ergonomics*, **20** (2014), no. 3, 525-533. http://dx.doi.org/10.1080/10803548.2014.11077060

[4] H. A. Khande, P. A. Narwade, Structural Analysis of Two Wheeler Handlebar, *International Journal of Advanced Science and Technology*, **68** (2014), 65-76. http://dx.doi.org/10.14257/ijast.2014.68.07

[5] J. P. Karthik, K. L. Chaitanya, C. Tara Sasanka, Fatigue Life Prediction of a Parabolic Spring under Non-constant Amplitude Proportional Loading Using Finite Element Method, *International Journal of Advanced Science and Technology*, **46** (2012), 143-156.

[6] M. H. Sigari, M. R. Pourshahabi, M. Soryani, M. Fathy, A Review on Driver Face Monitoring Systems for Fatigue and Distraction Detection, *International Journal of Advanced Science and Technology*, **64** (2014), 73-100. http://dx.doi.org/10.14257/ijast.2014.64.07

[7] E. Acar, K. Solanki, Improving the accuracy of vehicle crashworthiness response predictions using an ensemble of metamodels, *International Journal of Crashworthiness*, **14** (2009), no. 1, 49-61. http://dx.doi.org/10.1080/13588260802462419

[8] J. U. Cho, A Study on Safety Design through Static Analysis at Car Seat Frame, *Advanced Science and Technology Letters*, **130** (2016), 34-38. http://dx.doi.org/10.14257/astl.2016.130.08

Received: May 1, 2016; Published: August 15, 2016