International Journal of Advanced Smart Convergence Vol.11 No.4 266-272 (2022)
http://dx.doi.org/10.7236/IJASC.2022.11.4.266

IJASC 22-4-30

Verification of Stress Analysis on the Bracket of Bus Bear Chassis

Kim, Gyu Sung

Professor, Department of Smart Automotive Engineering, Seoil University, Korea
Koreagskim@seoil.ac.kr

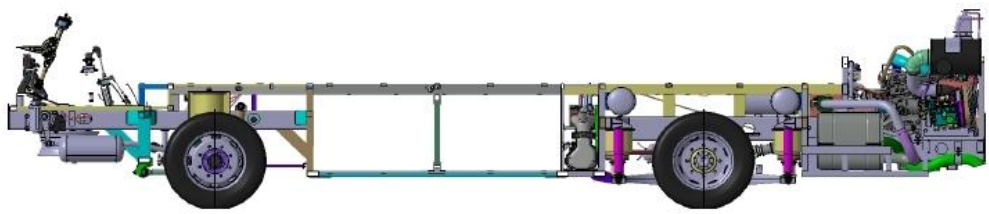
Abstract

Structural stress analysis is performed to confirm the safety of the structures before the construction, and stress analysis is performed to evaluate the safety of various components before the ship or vehicle corresponding to the moving structure is manufactured. In this case, the stress analysis work is performed using the stress analysis software of each company. The results of the stress analysis based on the boundary conditions of the applied loads are analyzed to evaluate the safety of the structure, but the results are difficult to verify because most of the stress analysis software possessed by each company is one. In this paper, we were performed the stress analysis of the bracket applied to the bare chassis of the 30-passenger bus under development is performed by HYPERMESH. In order to verify this, the stress analysis is performed using ANSA/META under the same boundary condition. The stress analysis results of ANSA/META and HYPERMESH showed that they had the same stress distribution and the maximum stress occurred at the same location. Taken together, the results of stress analysis using HYPERMESH were reliable.

Keywords: *Stress analysis, Bracket, Bus, Boundary condition, Bare chassis*

1. Introduction

The bare chassis of the 30-passenger bus in the development stage is as shown in Figure 1. The bare chassis is a basic frame constituting the vehicle body, and is a space in which an engine and a mission necessary for the operation of the bus are installed[1, 2].



Manuscript Received: December. 2, 2022 / Revised: December. 5, 2022 / Accepted: December. 7, 2022
Corresponding Author: gskim@seoil.ac.kr
Tel:*** - **** - ****
Professor, Department of Smart Automotive Engineering, Seoil University, Korea

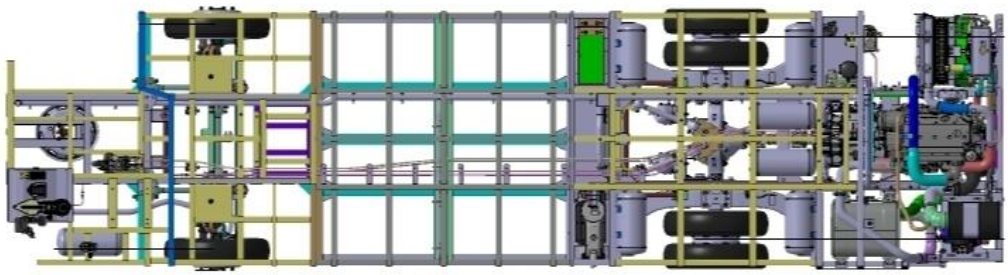


Figure 1. Bare chassis of developing bus

The space loading man or the load at the upper part of the bare chassis of the bus is installed[3-5]. The bracket to do the stress analysis is positioned in the bare chassis[6-8]. And it is the same as that of Figure 2. The brackets 1 and 2 are connected to the air mounting and the shock absorber to damp the load of the vehicle body on the upper part, and the load is applied to the front frame when braking or accelerating[9, 10]. In this paper, stress analysis is performed on two types of bracket in Figure 2.[1, 2]

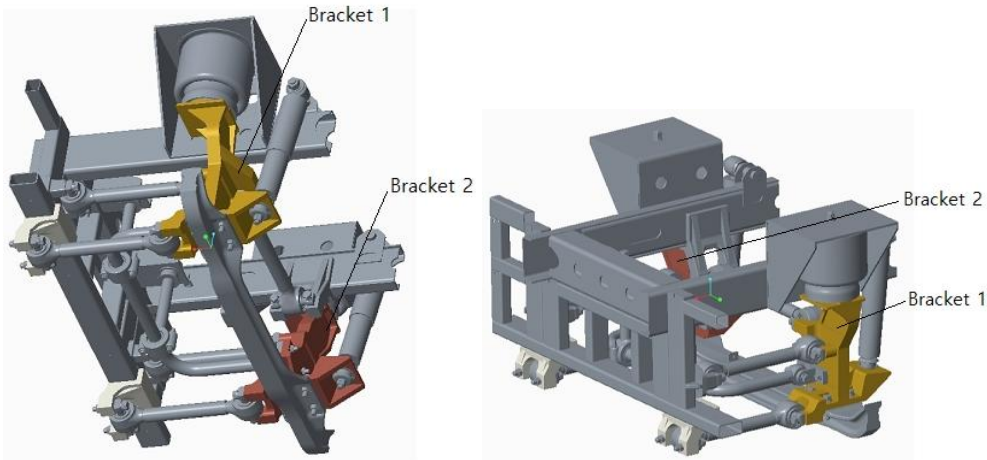


Figure 2. Positions of brackets

In order to perform the stress analysis, the material properties of the constituent materials must be specified. The bracket material to be analyzed for stress is SC450, and the mechanical properties are shown in Table 1.

Table 1. Mechanical property of the SC450

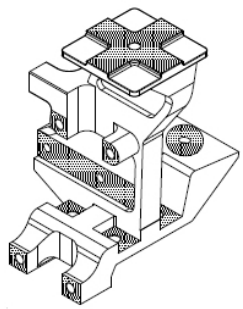
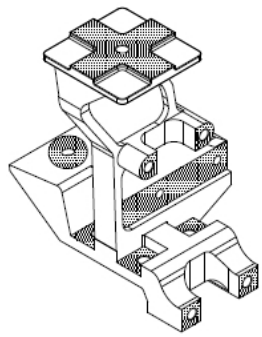
Sort	SC450
Coefficient of elasticity[GPa]	200
Poisson's ratio	0.29
Fatigue limit[MPa]	275
Yield limit[MPa]	350-550
Tensile strength[MPa]	650-880
Elongation ratio[%]	8-25

In this paper, the stress analysis of the bracket applied to the bare chassis of the 30-passenger bus under development is performed by HYPERMESH. To verify this, ANSA/META is used under the same boundary condition.

2. Modeling

The modeling shape applied to HYPERMESH, a software for structural analysis, and the modeling shape applied to ANSA/META to verify this are shown in Table 2 for stress analysis of brackets constructed in a 30-passenger bus bare chassis.

Table 2. Modeling diagram of HYPERMESH and ANSA/META

	HYPERMESH	ANSA/META
Bracket 1		
Bracket 2		

The finite element model using HYPERMESH was used by Altair's HYPERMESH 2017, and the solver used for structural analysis was Altair's Optistruct. The finite element model information applied to HYPERMESH is shown in Table 3[1].

Table 3. Finite element model information of HYPERMESH

Classification	Mesh No.	Mesh pattern
Bracket 1	665,187	Tetra
Bracket 2	601,303	Tetra

The finite element model information applied to ANSA/META is shown in Table 4.

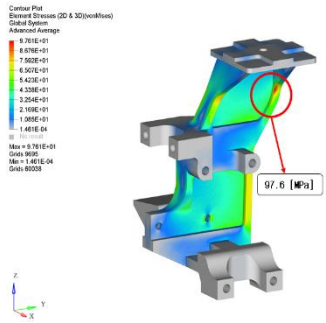
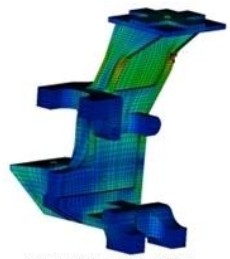
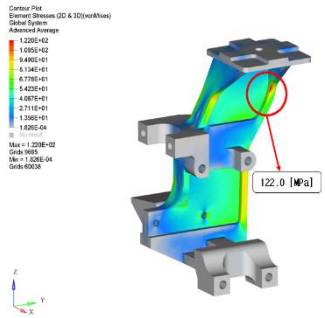
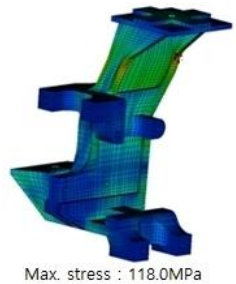
Table 4. Finite element model information of ANSA/META

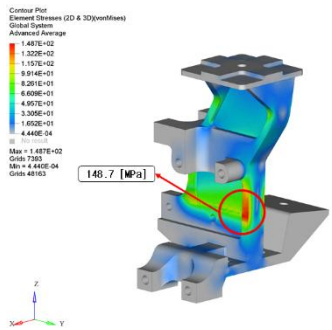
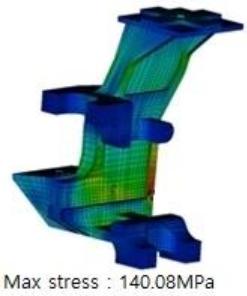
Used tool	Bracket No.	Mesh No.
Solver : EPILYSIS	Bracket 1	Hexas : 21921
Pre : ANSA v21.0.0 Post : META v21.0.0	Bracket 2	Hexas : 18053

3. Stress analysis comparison

In order to compare HYPERMESH and ANSA/META, which are the software used for stress analysis, the results of stress analysis for Bracket 1 under the same boundary conditions are shown in Table 5. In the analysis of the load of the boundary condition applied to the bracket, the effect of the load of the vehicle body attenuating through the air mounting and the shock absorber was not considered, and the situation where the load is applied to the bracket as it is was considered to apply the severe condition. In Table 5, No. 1 is the result of stress analysis for Bracket 1 when the load of the vehicle body is 20kN when the vehicle is stopped or flat travel, and No. 2 is the result of stress analysis when the load of the vehicle body is 25kN in the same form as No. 1. No. 3 is the result of stress analysis applied to Bracket 1 when the load of the vehicle body is 20kN and the braking force of 10kN is applied to each of the four links at the same time.

Table 5. Stress analysis results of the HYPERMESH and ANSA/META for Bracket 1

No.	HYPERMESH	ANSA/META
1		
2		

3		 Max stress : 140.08MPa

In Table 6, No. 4 is the result of stress analysis for Bracket 2 when the load of the vehicle body is 20kN. No.5 is the result of the stress analysis when the vehicle passes through the unevenness and 25kN acts on the lower part and 15kN acts on each of the four links while braking. No.6 is the result of the stress analysis when the body load is 25kN and the braking is 15kN in each of the four links at the same time.

Table 6. Stress analysis results of the HYPERMESH and ANSA/META for Bracket 2

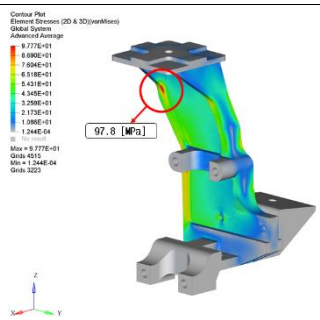
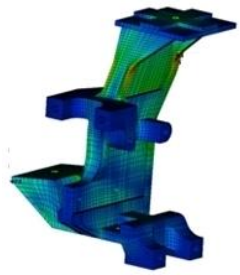
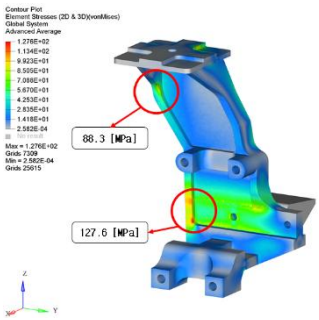
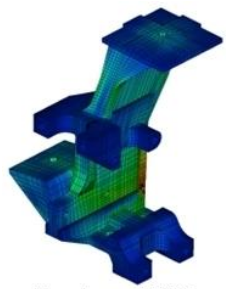
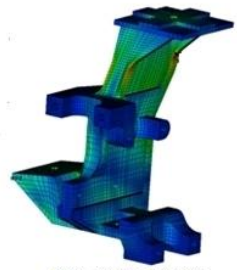
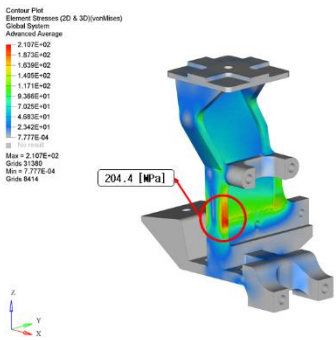
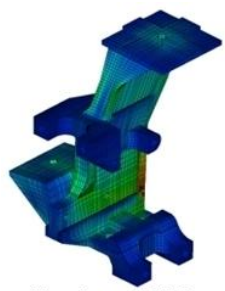
No.	HYPERMESH	ANSA/META
4		 Max. stress : 97.1MPa
5		 Max. stress : 122.8MPa  Max. stress : 87.9MPa
6		 Max. stress : 196.4MPa

Table 5, which is the comparison result of stress analysis for Bracket 1, and Table 6, which is the comparison result of stress analysis for Bracket 2, are summarized and the error of maximum stress is shown as Table 7.

Table 7. Stress analysis results of the HYPERMESH and ANSA/META for Bracket 2

No.	HYPERMESH Max. Stress[MPa]	ANSA/META Max. Stress[MPa]	Max. Stress Difference(%)	Max. Stress Position
1	97.6	92.7	5	Same
2	122.0	118.0	3.3	Same
3	148.7	140.08	5.8	Same
4	97.8	97.1	0.7	Same
5	127.6	122.8	3.8	Same
6	204.4	196.4	3.9	Same

Comparing the results of two software, HYPERMESH and ANSA/META, which were applied to perform the stress analysis for the bracket, the maximum stress has an error of about 3.7% on average, but the stress distribution is generally consistent and the maximum stress position is the same. Based on this evaluation, it is judged that the stress analysis by HYPERMESH can be trusted.

4. Conclusion

In this paper, we were verified that the stress analysis was performed on the bracket applied to the bare chassis of the 30-passenger bus in the development stage. The reason for the stress analysis is to evaluate the stability of the proposed bracket in the situation where the load is repeatedly applied to the bracket while the bus is running. The software used for the stress analysis is HYPERMESH and ANSA/META. The modeling for stress analysis was performed using a given bracket shape, and the boundary condition was given a severe condition in which the load of the vehicle body was transferred to the bracket. As a result of comparing and verifying using two kinds of stress analysis software, HYPERMESH and ANSA/META, the following conclusions were obtained.

1) The maximum stress of two software, HYPERMESH and ANSA/META, has an error of 3.7% on average,
2) The stress distribution of two software, HYPERMESH and ANSA/META, is consistent.
3) The maximum stress positions of HYPERMESH and ANSA/META software are the same.

Acknowledgement

This work was supported by Seoil University Academic Research Fund in 2022.

References

[1] Wan-Mug Choi, “Safety Evaluate of Brackets for Bare Chassis of a 30-seated Bus,” International Journal of Advanced Smart Convergence Vol.11 No.3 215-221, DOI: http://dx.doi.org/10.7236/IJASC.2022.11.3.215, 2022

[2] Yang Seung Bok, “Boundary Condition for Bare Chassis Brackets of the Commercial Vehicle,” International Journal of Advanced Smart Convergence Vol.11 No.1 94-100, DOI: http://dx.doi.org/10.7236/IJASC.2022.11.1.94, 2022

[3] D. H. Lee & J. U. Cho, “Convergence Study on Damage of the Bonded Part at TDCB Structure with the Laminate Angle Manufactured with CFRP,” Journal of the Korea Convergence Society, 9(12), 175-180. DOI:http://dx.doi.org/10.15207/JKCS.2018.9.12.175, 2018

[4] Y. K. Park & B. G. Lee, “A Study on the Structural Analysis of the Spindle of Swiss Turn Type Lathe for Ultra Precision Convergence Machining,” Journal of the Korea Convergence Society, 9(5), 145-150. DOI:http://dx.doi.org/10.15207/JKCS.2018.9.5.145, 2018

[5] Jin-soo Park, Hyun-jin Park, Byung-ki Oh, “Optimization of Bearing Bracket for Automotive Drive Shaft,” Fall Conference Proceedings of The Korean Society Of Automotive Engineers, pp. 408-409, 2018. 11

[6] Jong-chan Kim, Hyun-gu Ji, “Bracket Topology Optimization for Robust Design in Concept Design,” Fall Conference Proceedings of The Korean Society Of Automotive Engineers, pp. 810-810, 2018. 11

[7] Y. T. Oh, “Study on the Structural Analysis of SCARA Industrial Robot,” Journal of the Korean Society of Mechanical Technology, 19(6), 866-871. DOI: http://dx.doi.org/10.17958/ksmt.19.6.201712.866, 2017

[8] J. H. Lee & J. U. Cho, “A Study on Convergence Technique due to the Shape of Cruiser Board through Structural Analysis,” Journal of the Korea Convergence Society, 6(4), 99-105. DOI:http://dx.doi.org/10.15207/jkcs.2015.6.4.099, 2015

[9] Hong-suk Chang, Jung-sup Lee, “Design of chassis mounting brackets using optimization method,” Spring conferences Proceedings of The Korean Society Of Automotive Engineers, pp. 1185-1190, 2012. 5

[10] B. Hwang, H. Lee, J. Jang and I. Joo, "A Study of Stress Distribution of a pulley for automobile at static load condition," Fall Conference Proceedings of Korean Society of Automotive Engineers, Vol.2, pp.1180~1184, 2003.