1. Introduction

The vehicles accounts for a large portion of domestic cargo transportation by land. Large trucks which are running the long-distance at high speeds play a big role in land transport. The fuel efficiency can be improved by reducing the air resistance. So, the small efficiency improvements can save a lot of money in a gas-guzzling car. By using the flow analysis on large truck as well as passenger car, many researches at Europe are being actively performed about the reduction of air resistance. In case of large trucks, the deflectors are installed mainly on the top of cap where the seat of driver is located for the purpose of improving the aerodynamic performance to reduce air resistance[1-5]. In this study, the overall states of air flow under the condition that the truck with or without side pairing is driving at a maximum speed of 90 km/h

# Corresponding Author : jucho@kongju.ac.kr
Tel: +82-41-521-9271, Fax:+82-41-555-9123
regulated by domestic law are investigated through CFD numerical analysis\textsuperscript{[6-11]}. The drag by flow affects the recovery of pressure from the rear of the truck body\textsuperscript{[8-11]}. The results of this study are thought to be the effective data at improving performance by reducing the resistance against the airflow flown from the truck in driving itself.

2. Study Models and Boundary Conditions

2.1 Study models

In this study, each of the three-dimensional models on trucks is shown by Figs. 1 (a), (b) and (c), respectively. In this study, the truck at Fig. 1 (a) was modeled with no side-pairing. The truck at Fig. 1(b) was modeled with a shock absorber bar installed at the truck which is commonly driven on current roads rather than side-pairing. Finally, the truck at Fig. 1(c) was modeled with the side-pairing to cover the sidewall of the truck's body as much as possible. Table 1 shows the meshes of models 1, 2 and 3.

2.2 Boundary conditions of models

Flow analyses in this study were carried out by using the CFX program of ANSYS. Figs. 2 (a), (b) and (c) show the boundary conditions specified in fluid models 1, 2 and 3, respectively. The flow rate of inlet is designated as 90km/h(25m/s). And the temperature of air flow is 25°C.

3. Flow Analysis Results

3.1 Flow analysis result of model 1
Figs. 3, 4 and 5 show the pressure contours and flow rates on the rear of the truck at model 1 with no side fairing. As shown by Fig. 3, the maximum pressure on the rear of truck is $7.629 \times 10^{-6} Pa$. At Fig. 4, the maximum flow rate of $39.08 m/s$ is shown around the truck. Also, Fig. 5 shows the fastest flow rate passing through the top of the truck. By using the function calculator of CFX-Post, the drag ($F_d$) acting on the truck body became $-25.1686 N$ at model 1. As the front section area of the truck was $0.111 m^2$, the drag coefficient of the truck was 0.56116.

3.2 Flow analysis result of model 2

Figs. 6, 7 and 8 show the pressure contours and flow rates on the rear of the truck at model 2 equipped with a shock absorber bar. As shown by Fig. 6, the maximum pressure on the rear of truck is $8.306 \times 10^{-2} Pa$. At Fig. 7, the maximum flow rate of $39.49 m/s$ is shown around the truck. Also, Fig. 8 shows the fastest flow rate passing through the top of the truck. By using the function calculator of CFX-Post, the drag ($F_d$) acting on the truck body became $-25.1831 N$ at model 2. As the front section area of the truck was $0.111 m^2$, the drag coefficient of the truck was 0.56148.
3.3 Flow analysis result of model 3

Figs. 9, 10 and 11 show the pressure contours and flow rates on the rear of the truck at model 3 installed with a side-fairing. As shown by Fig. 9, the maximum pressure on the rear of truck is 1.317 × 10⁻¹ Pa. At Fig. 10, the maximum flow rate of 39.64 m/s is shown around the truck. Also, Fig. 11 shows the fastest flow rate passing through the top of the truck. By using the function calculator of CFX-Post, the drag \( F_d \) acting on the truck body became \(-26.6487\) N at model 3. As the front section area of the truck was 0.111 m², the drag coefficient of the truck was 0.59416.
3.4 Overall analysis results of models 1, 2 and 3

Figs. 12, 13 and 14 show the pressure contours on the middle plane of the truck body at models 1, 2 and 3, respectively. And Figs. 15, 16, 17 show the streamlines that represent the velocity of air flowing around the body of truck at models 1, 2 and 3, respectively. As shown by these figures, the air flowing down the truck is not going out the body to follow the underside of truck body to the end until the rear of the truck. Instead, as the air escapes to the side of a truck, the pressure appears high near the truck’s side of the lower ground.
4. Conclusion

In this study, the overall states of air flow under the condition that the truck with or without side pairing is driving at a maximum speed of 90 km/h regulated by domestic law are investigated through CFD numerical analysis. The study results are as follows;

1. At model 3, the maximum pressure of $1.317 \times 10^{-1} Pa$ at the rear of the truck was shown to be the greatest among the three models. And the maximum flow rate of $39.64 m/s$ at model 3 was the largest among three models.

2. The drag with drag coefficient at model 3 was clearly higher in comparison to those of the other two models.

3. At all models, it can be seen that the air flow goes under the body of truck. This air does not flow along the underside of truck to the rear on the body of truck but through the sides of the underside.

4. The results of this study are thought to be the effective data at improving performance by reducing resistance against the airflow flown from the truck in driving itself.

REFERENCES

1. Oh, B. S., and Cho, J. U., “A Convergence Study through Flow Analysis due to the Configuration of Automotive Air Breather”, Journal of the Korea Convergence Society, Vol. 9, No. 10, pp. 265-270, 2018.

2. Jung, S. H., “The Control of Spring-Mass-Damper Convergence System using $H\infty$ Controller and $\mu$-Synthesis Controller”, Journal of the Korea Convergence Society, Vol. 8, No. 5, pp. 1-11, 2017.

3. Cho, J. U., “A Flow Analysis on Wing Shape of Cooling Fan at Automobile”, Journal of the Korea Convergence Society, Vol. 5, No. 4, pp. 75-79, 2014.

4. Ahn, K. C., Kim, B. H. and Lee, C. W., “Flow Analysis of Flowmeter for High-Viscosity Paint Spraying System,” Journal of the Korean Society of Mechanical Technology, Vol. 16, No. 4, pp. 1709-1714, 2014.

5. Lee, C. R. and Kim, B. H., “Flow Analysis of Cylindrical Helical Water Turbine for Micro Hydro-power”, Journal of the Korean Society of Mechanical Technology, Vol. 20, No. 2, pp. 187-193, 2018.

6. Kim, G. H. and Lee, D. R., “A Study on The Flow Analysis in the Turbocharger Turbine with Different Rotational speeds”, Journal of the Korean Society of Mechanical Technology, Vol. 18, No. 5, pp. 734-739, 2016.

7. Lee, J. B., Yang, D. H., Kim, S. Y. and Kim, K. S., “A Study on Flow Analysis of the Blast Valve for Anti-aircraft Shelter”, Journal of the Korean Society of Mechanical Technology, Vol. 17, No. 5, pp. 1107-1115, 2015.

8. Cho, J. Y., Go, S. H. and Kim, H. G., “Analysis of Drainage Efficiency of Different Type of Drainage using Computational Fluid Dynamic Method,” Journal of the Korean Society of
Manufacturing Process Engineers, Vol. 16, No. 2, pp. 34-43, 2017.

9. Kim, H. J. and Lee, Y. Min., “Experimental Study on Air Flow Characteristics of Axial Dual-blade Fan,” Journal of the Korean Society of Manufacturing Process Engineers, Vol. 13, No. 4, pp. 113-120, 2014.

10. Lee, D. H., Park, S. S., Ko, T. J. and Shim, J. S., “Effect of the Texture Shape Aspect Ratio on Friction Reduction in a Hydrodynamic Lubrication Regime,” Journal of the Korean Society of Manufacturing Process Engineers, Vol. 16, No. 2, pp. 63-68, 2017.

11. Jun, C. W., Sohn, J. H. and Yang, M. S., “Comparison of Fluid Modeling Methods Based on SPH and ISPH for a Buoy Design for a Wave Energy Converter,” Journal of the Korean Society of Manufacturing Process Engineers, Vol. 16, No. 3, pp. 94-99, 2017.