한국기계가공학회지, 제19권, 제9호, pp. 9～14(2020.09) ISSN 1598-6721(Print)
Journal of the Korean Society of Manufacturing Process Engineers, Vol. 19, No. 9, pp. 9～14(2020.09) ISSN 2288-0771(Online)

https://doi.org/10.14775/ksmpe.2020.19.09.009

Flow Analysis due to the Slant Angle of a Windscreen at the Front of a Car Body

Kyekwang Choi*, Jaeung Cho**,#

*Department of Metal Mold Design Engineering, Kongju National UNIV.
**Division of Mechanical and Automotive Engineering, Kongju National UNIV.

차체 전방의 앞 유리 경사각도에 따른 유동해석에 관한 연구

최계광*, 조재웅**,#

*공주대학교 금형설계공학과, **공주대학교 기계자동차공학부

(Received 16 June 2020; received in revised form 20 June 2020; accepted 06 July 2020)

ABSTRACT

In this study, CFX analyses were performed with flow models to minimize the flow resistance due to the windscreen on the front of a car body. The results indicated that the greater the slant angle of the windshield, the greater the maximum pressure area. The lower the slant angle of the windscreen, the smaller the area in which the air collides with the front of the car body and the more smoothly the air moves. The results of this study can be applied to increase fuel economy under driving conditions by changing the slant angle of the vehicle's windscreen.

Keywords : Front Car Body(차체 전방), Windscreen(앞유리), Slant Angle(경사각도), Flow Analysis(유동해석)

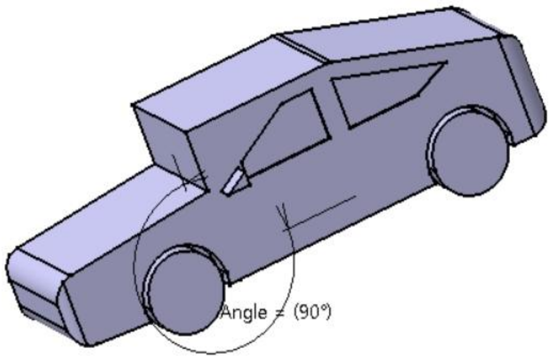
1. Introduction

The fuel economy measured currently differs greatly from the case in which the actual driver drives on the road. If the fuel economy is measured by including the condition of sloping road, it can be predicted more closely in the real state. Many studies are being conducted in order to improve the fuel economy of cars[1-4]. In this study, the flow analysis was performed with the air flown near windscreen of the front car body that affected the fuel efficiency[5-7]. The effects of air resistances on the vehicles at the driving speed of 90 km/h were analyzed with three kinds of vehicle models designed by the CATIA program, due to the shape of the vehicle's front glass angle. In fact, it takes the expensive cost at conducting a flow experiment because the experiment equipment on car body, the personnel and the time are required. But the flow resistance and streamline can be investigated and evaluated by only a flow analysis of CFX[8-9]. So, the design evaluation of windscreen on the front car body was carried out in order to minimize the flow resistance in this study.

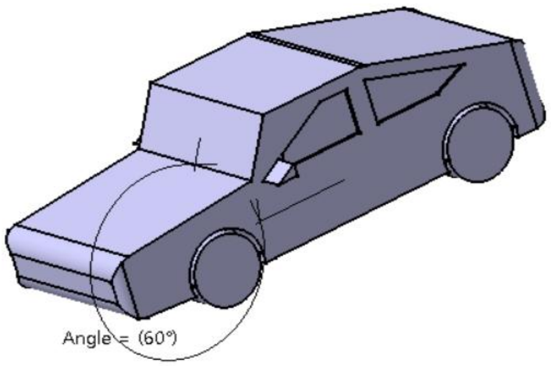
Corresponding Author : jucho@kongju.ac.kr
Tel: +82-41-521-9271, Fax:+82-41-555-9123

2. Study Models and Boundary Conditions

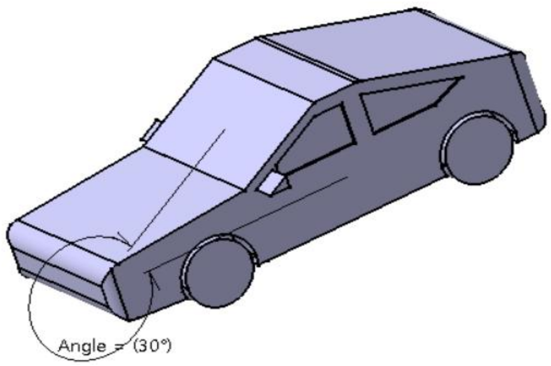
The study models with windscreens on the front car body were reduced them to one-eighth of the size of a real car and three car models were designed with CATIA program. The shapes of all models are shown at Fig. 1.



(a) Model 1

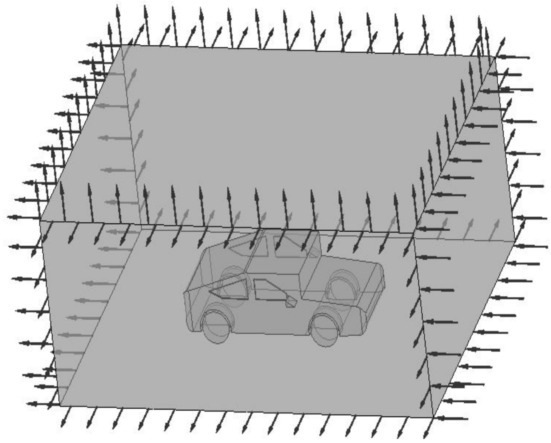


(b) model 2

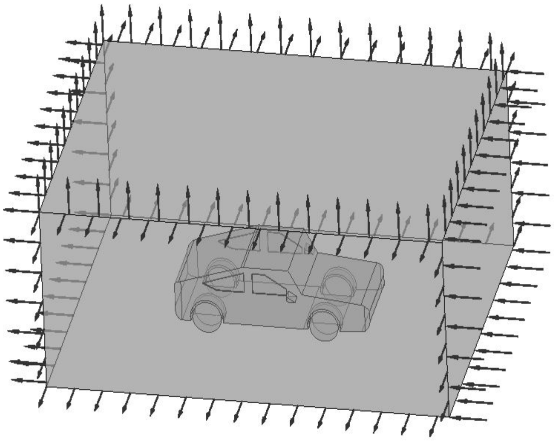


(c) Model 3

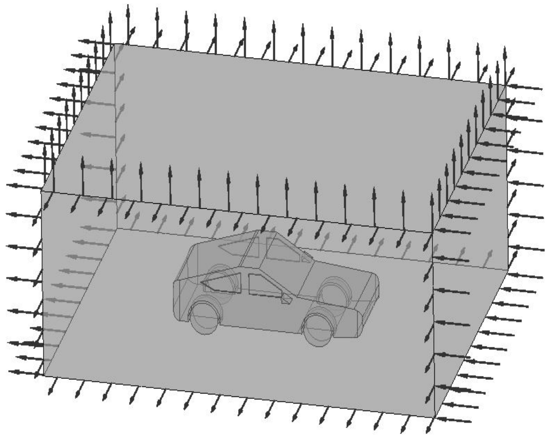
Fig. 1 Shapes of study models due to slant angles



(a) Model 1



(b) Model 2



(c) Model 3

Fig. 2 Boundary conditions of flow models

Table 1 Mesh information of models

	Nodes	Elements
Model 1	17194	94725
Model 2	17220	94882
Model 3	17399	95926

Table 2 Property of air flow model

Temperature	25℃
Pressure	0Pa
Density	1.185kg/m^3

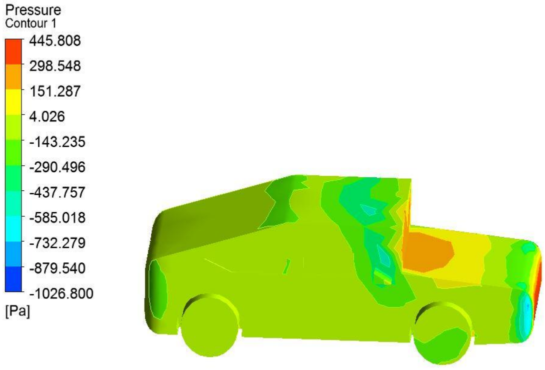
The front windscreens in cases of models 1, 2 and 3 were set at the slant angles of 90°, 60° and 30° to the horizontal plane, respectively. As shown by Fig. 2, A quadrilateral box is formed near the solid model and the flow model is made by removing this solid model from the box. This square box has the dimensions of 500mm, 250mm and 500mm in length, height and width for each model.

Also, Fig. 2 shows the boundary conditions of flow models. The wind speed of 90km/h was applied at the entrance as the inlet condition and the atmosphere pressure of 0Pa was set at the outlet. The lower foor was fixed as wall condition. Table 1 shows the mesh informations of flow models. As the fluid property of air, the flow models were shown to have the temperature, pressure, and density at Table 2.

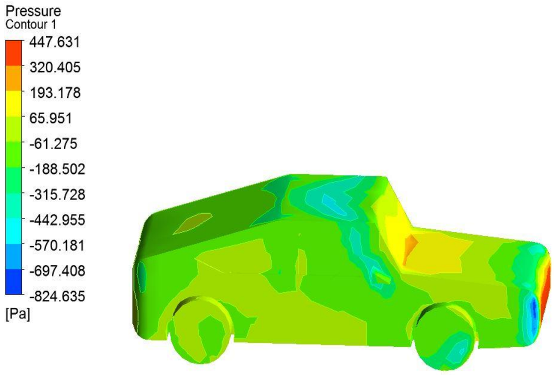
3. Study Results

Fig. 3 shows the pressure contours on car body surfaces at all models. Maximum pressures at models 1, 2 and 3 can be shown to be the highest in the front of the car body.

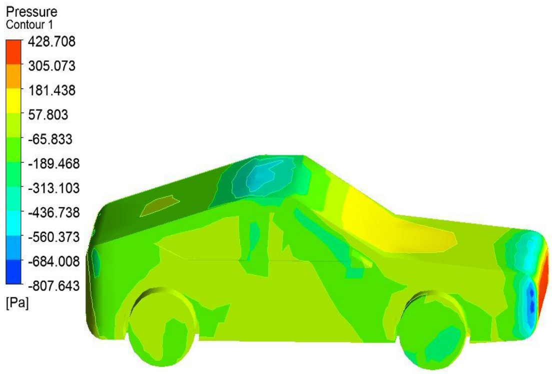
The greater slant angle of the windshield, the greater area of maximum pressure. Generally, model 1 among three models is shown to have the widest area with higher pressure of about 298 Pa in the front of the car body. Fig. 4 shows the contours of air flow rates on the side plane for each model. b



(a) Model 1



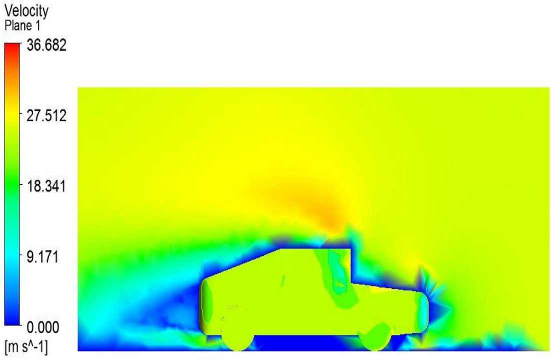
(b) Model 2



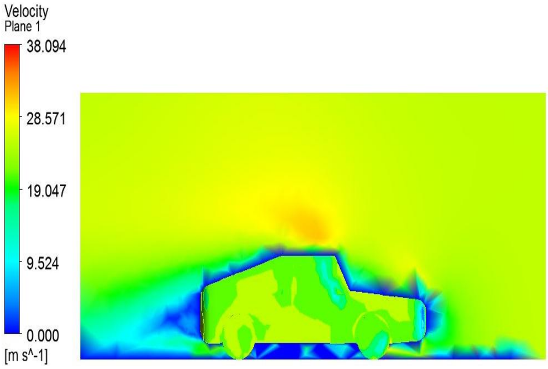
(c) Model 3

Fig. 3 Pressure contours on car body surfaces

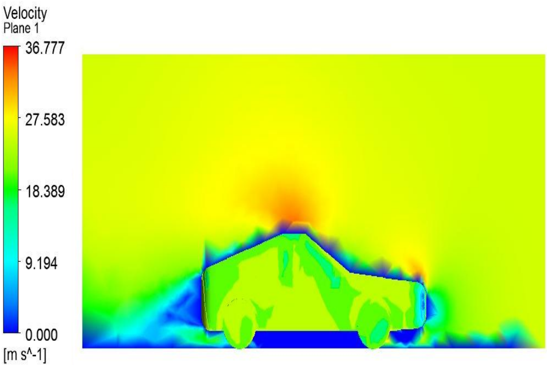
The maximum flow rates of 36.682m/s, 38.094m/s and 36.777m/s were shown at the initial starting point of the car roof in cases of model 1, model 2 and model 3, respectively.



(a) Model 1



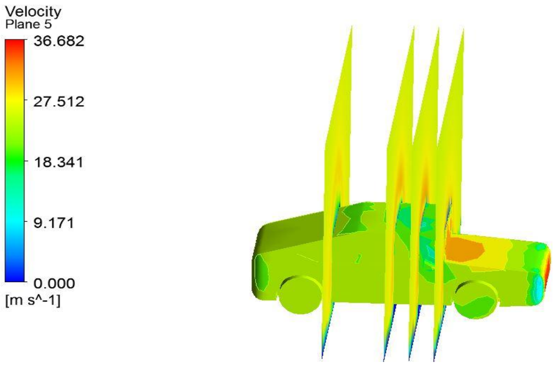
(b) Model 2



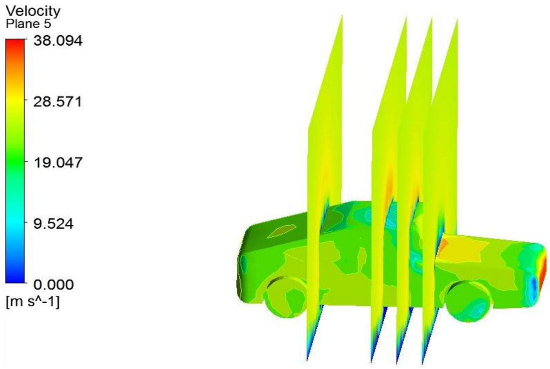
(c) Model 3

Fig. 4 Contours of flow rates on side planes of models

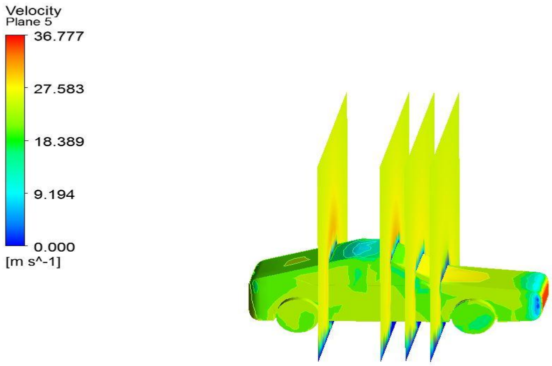
Models 1, 2 and 3 have similar rates at the rear of the vehicle, but show a difference in the amount of flow rate due to the slant angle of windscreen at the front of the vehicle's body.



(a) Model 1



(b) Model 2

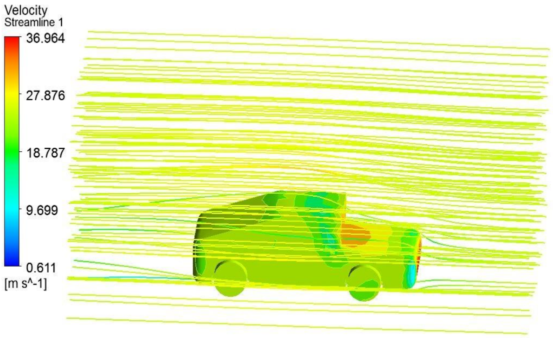


(c) Model 3

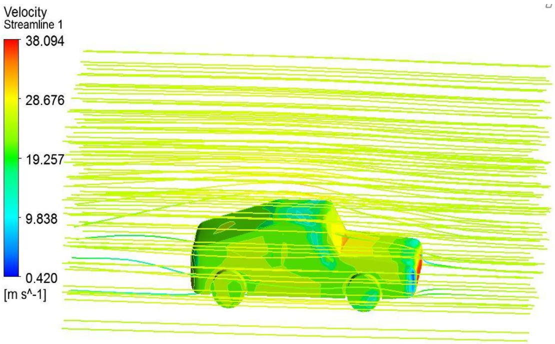
Fig. 5 Contours of flow rates on middle planes of models

Fig. 5 shows the contours of flow rates at the middle planes for each model. The maximum flow velocities of 36.682m/s, 38.094m/s and 36.777m/s were shown at the initial starting point of the car roof in cases of model 1, model 2 and model 3, respectively. It can be shown that the lower the slant angle of windscreen, the smaller the flow rate goes over the car.

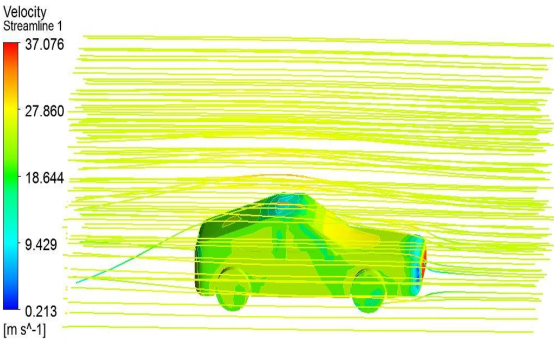
Fig, 6 shows the contours of flow streamlines for each model. The lower the slant angle of windscreen, the smaller the area in which the air collides with the front car body and the more smoothly the air moves over. On the other hand, the air that fails to move smoothly remains intact, thus tends to disrupt the speed of the car.



(a) Model 1



(b) Model 2



(c) Model 3

Fig. 6 Contours of flow streamlines for each model

4. Conclusion

In this study, CFX analyses were performed with the flow models on three kinds of vehicle models due to windscreen on the front car body in order to minimize the flow resistance. The study results are as follows;

1. Maximum pressures at models 1, 2 and 3 can be shown to be the highest in the front of the car body. The greater slant angle of the windshield, the greater area of maximum pressure.
2. Models 1, 2 and 3 have similar speeds at the rear of the vehicle, but show a difference in the amount of flow rate due to the slant angle of windscreen at the front of the vehicle's body.
3. The lower the slant angle of windscreen, the smaller the area in which the air collides with the front car body and the more smoothly the air moves over. Therefore, it is considered that the results of this study can be applied to increase fuel economy under driving by changing the slant angle of windscreen at the front car body.

REFERENCES

1. Choi, C. H, Noh, J. G. and Kim, J. H., “Numerical Simulation of Cavitating Flow Around Turbopump Inducer,” The KSFM Journal of Fluid Machinery, Vol. 13, No. 3, pp 49-53, 2010.
2. Jo, S. H, Park, J. I. and Nam, K. W., “Numerical Simulation in the IC Engine Lubricating Gerotor Oil Pump,” Transactions of the Korean Society of Mechanical Engineers B, Vol. 30, No. 10, pp. 1019-1025, 2006.
3. Lee, S. H. and Hur, N. K., "Numerical Study on

Effects of Design Factors on Flow Characteristics of a Vane Pump," The KSFM Journal of Fluid Machinery, Vol. 10, No. 6, pp. 24-31, 2007.
4. Kim, H. J. and Lee, Y. M., "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.
5. Oh, H. J., Lyu, J. H. and Park, J. Y., "Rodent Experiments for Pedestrian Flow Simulation at Exit with Various Angles," Journal of the Korean Society of Manufacturing Process Engineers, Vol. 15, No. 4, pp. 30-39, 2016.
6. Choi, K. K., Hong, S. M. and Han, S. R., "Effect of Flow Pattern of Coolant for Injection Mold on the Deformation of Injection Molding," Journal of the Korean Society of Manufacturing Process Engineers, Vol. 14, No. 4, pp. 92-99, 2015.
7. Cho, H. J., "Potential Flow Analysis around a Inlet Duct of a Water-jet Propulsion System," Journal of the Korean Society of Mechanical Technology, Vol. 16, No. 1, pp. 1149-1154, 2014.
8. Kim, H. K. and Cho, H. M., "A Study on Comparative Analysis of Internal ECV Through Flow Analysis," Journal of the Korean Society of Mechanical Technology, Vol. 16, No. 1, pp. 1161-1166, 2014.
9. Lee, H. C. and Cho, J. U., "A Study on Air Flow Analysis due to the Shape of Automotive Body", Journal of the Korea Convergence Society, Vol. 5, No. 2, pp. 19-23, 2014.