RES MILITARIS
REVUE EUROPEENNE D ETUDES EUROPEAN JOURNAL OF MILITARY STUDIES
Social Science Journal

Analysis of The Effect of Side Mirrors on Vehicle Aerodynamics

By

Ngoc Duong Hoang
Faculty of Automotive Engineering Technology, Industrial University of Ho Chi Minh City, Ho Chi Minh City 700000, Vietnam
Email: hoangngocduong@iuh.edu.vn

Khoi Nguyen Nguyen
Faculty of Automotive Engineering Technology, Industrial University of Ho Chi Minh City, Ho Chi Minh City 700000, Vietnam
Email: nguyenkhoinguyen@iuh.edu.vn

Son Tung Pham
Faculty of Automotive Engineering Technology, Industrial University of Ho Chi Minh City, Ho Chi Minh City 700000, Vietnam
Email: phamsontung@iuh.edu.vn

Abstract

During the operation of cars, aerodynamic drag is a factor that affects the stability, balance and working efficiency of the vehicle. Therefore, aerodynamic characteristics are always concerned by manufacturers in the world today in the design process to reduce the drag of the vehicle and that is the main goal of this article. Research using Solidworks to build a 3D model of the vehicle from the 2D drawing of the original sedan car, applying the finite element method (FEM) by Flow Simulation module to investigate the pressure and velocity distribution around the model. Then the shape of the side mirrors and tilt angle of vehicle A-Pillar were changed to reduce the vehicle's aerodynamic drag from comparing the aerodynamic characteristics of the two models. Simulation analysis shows that compared to the original model, the improved model has a reduced drag force of 4% and a drag coefficient of 7.1%. The results serve the research and design of vehicle models to ensure high efficiency of vehicle performance.

Keyworks: Side mirrors, Vehicle aerodynamics, Drag coefficient, Finite element method (FEM), Aerodynamics characteristic.

1. Introduction

Today, in addition to the development and improvement of cars in terms of engines and technology, changing the shape of the car is also interested in improving aerodynamics. From there, the designer can research and improve the car models to have a low coefficient of drag but still keep the unique features of that car. To better understand these effects, the researchers performed simulations on vehicle models similar to a real car through CFD software and applications. Typically, the studies in (Ayyagari et al., 2017; Tsai et al., 2009; Maji et al., 2021) focused on reducing drag coefficient by adding a spoiler or comparing two vehicles with and without a spoiler (Hu and Wong, 2011). Many research groups have used practical PIV experiments to analyze the influence of the rear tilt angle on the characteristics of the air flow around the vehicle (Tunay et al., 2016). In addition, many studies have been carried out on different types of vehicles such as buses, trucks, vans, passenger cars and many others. Specifically, the author (Kanekar et al., 2017) studied the bus drag coefficient when changing the bus shape design in order to reduce fuel consumption and emissions causing environmental pollution. As a result, the author concluded that with the entangled simulation model k-ε Realizable, when changing the bus shape compared to the original shape, the drag coefficient is reduced by 28% and fuel is saved by 20% when the bus is changed. moving at 80 km/h. The author (Muthuvel et al., 2013) has studied experimentally and simulated the aerodynamic influence on the exterior shape of the bus in order to reduce aerodynamic drag as well as reduce fuel consumption. As a result, the author concludes that when adjusting the design of the car's exterior shape compared to the original shape, aerodynamic drag is reduced by about 30% and significantly reduces fuel consumption. The author (Abinesh and Arunkumar, 2014; Yadav et al., 2017) used the CFD tool to analyze the effect of aerodynamic drag on fuel consumption on buses. These two studies have given the vehicle shape after adjustment, the authors both concluded that aerodynamic drag is reduced when adjusting the vehicle's shape. The authors in (Lakshmanan and Yadav, 2020) conducted aerodynamic analysis on sports cars, the effect of attaching the rear spoiler to reduce drag coefficient by building 3D models using Solidworks software, then CFD analysis of sports cars with and without a spoiler was performed by Ansys, the results showed that the coefficient of drag and drag of cars without a spoiler and with a spoiler were determined. determined. The graphs of velocity, pressure, and turbulence have also been shown in the article, and when attaching the spoiler, the drag coefficient was reduced by 11.4%. As the drag coefficient decreases, it improves fuel economy and vehicle stability. Therefore, the active spoiler helps to improve the aerodynamic performance of the car.

Although the field of automotive aerodynamics has been studied extensively around the world. However, corresponding to each different vehicle model, there are ways to improve the shape to reduce drag or different aerodynamic coefficients to ensure the design of the original car. Therefore, this study focuses on improving the shape of the two side mirrors by using Solidworks to build 3D models and compare the pressure and velocity distributions of the air flow around the model of the two vehicles to evaluate the aerodynamic characteristics of the vehicle after improvement.

2. The sedan vehicle model and the computational domain

The 2D and 3D sedan models are described in detail as shown in Fig. 1 with the total length L = 4580 mm, height H = 2005 mm, whole width W = 1447 mm on 3 axes of the x, y, z coordinate system along with the original dimensions of the side mirrors.

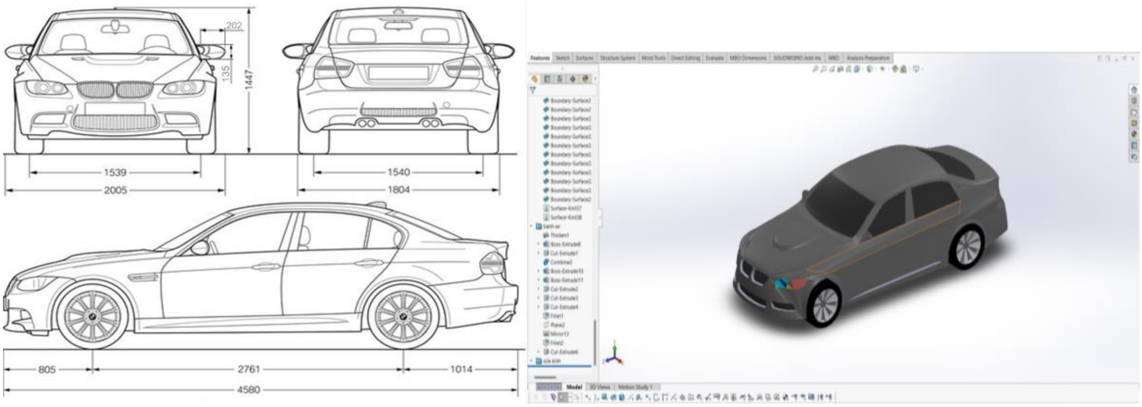


Figure 1. *The original sedan model (Model A)*

In Fig. 2, the sedan has improved the shape of the side mirror and the tilt angle of the A-pillar. This improvement only changes the mirror head angle of 60^0, the size does not change, the two sides of the A-pillar are tilted inward compared to the rear view mirror. with anterior glass plane of 30^0 similar to those studied (Zaareer and Mourad, 2022; Cheng et al., 2012; Al-Obaidi and Otten, 2018).

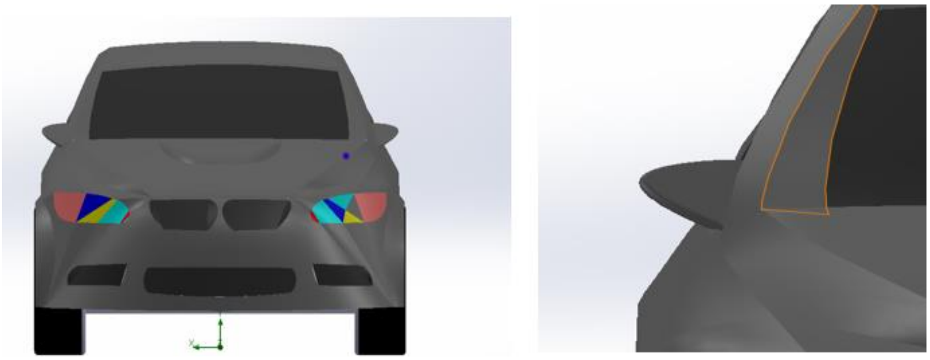


Figure 2. *The improved sedan model (Model B)*

The computational domain is the area of space surrounding the object that is limited during the simulation. The calculated domain must be large enough to be limited by planes to ensure that the air flow is not affected by the research model. However, it is also not possible to choose a computational domain that is too large, leading to wasting computer resources and increasing simulation time. Therefore, combined with the research works of the authors (Corallo et al., 2015; Tunay et al., 2014; Dang et al., 2016), the author proceeds to build the calculation domain with the parameters described in Fig. 3. In which, H is the vehicle height, W is the vehicle width, L is the vehicle length.

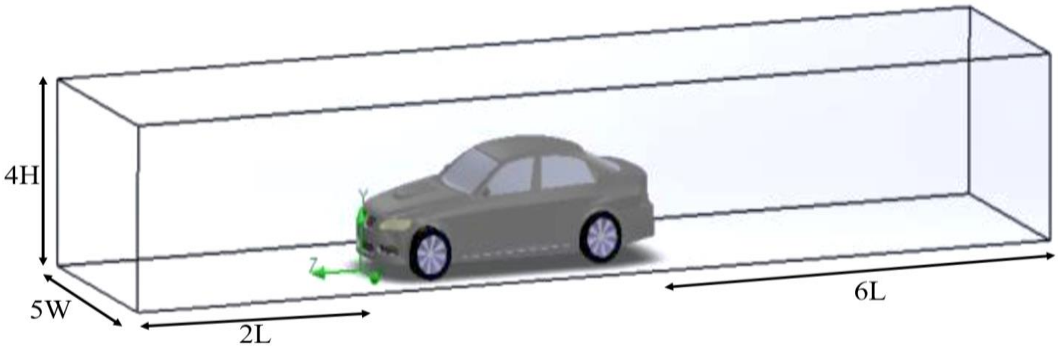


Figure 3. *The computational domain of model*

Meshing is the discretization of the simulation space into elements to perform numerical approximation. Grids come in two forms, structured and unstructured, each with their own strengths. In this study, we will use the mesh density at level 7 in the Flow Simulation module of Solidworks (Gukop et al., 2021) to ensure the accuracy of the results when simulating calculations. The complete meshed sedan model is shown in Fig. 4.

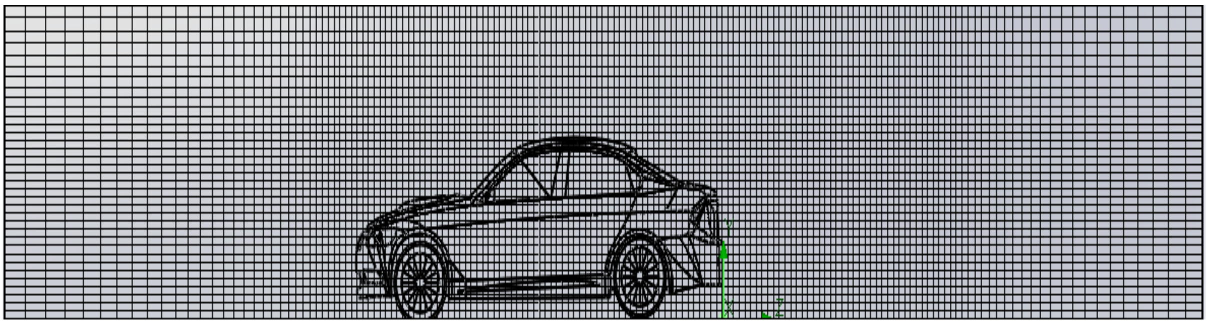


Figure 4. *The meshing model*

3. Analysis Results

The study uses the input air velocity value of 30 m/s, in the z direction. Therefore, the simulation results of the velocity distribution around the vehicle are as follows:

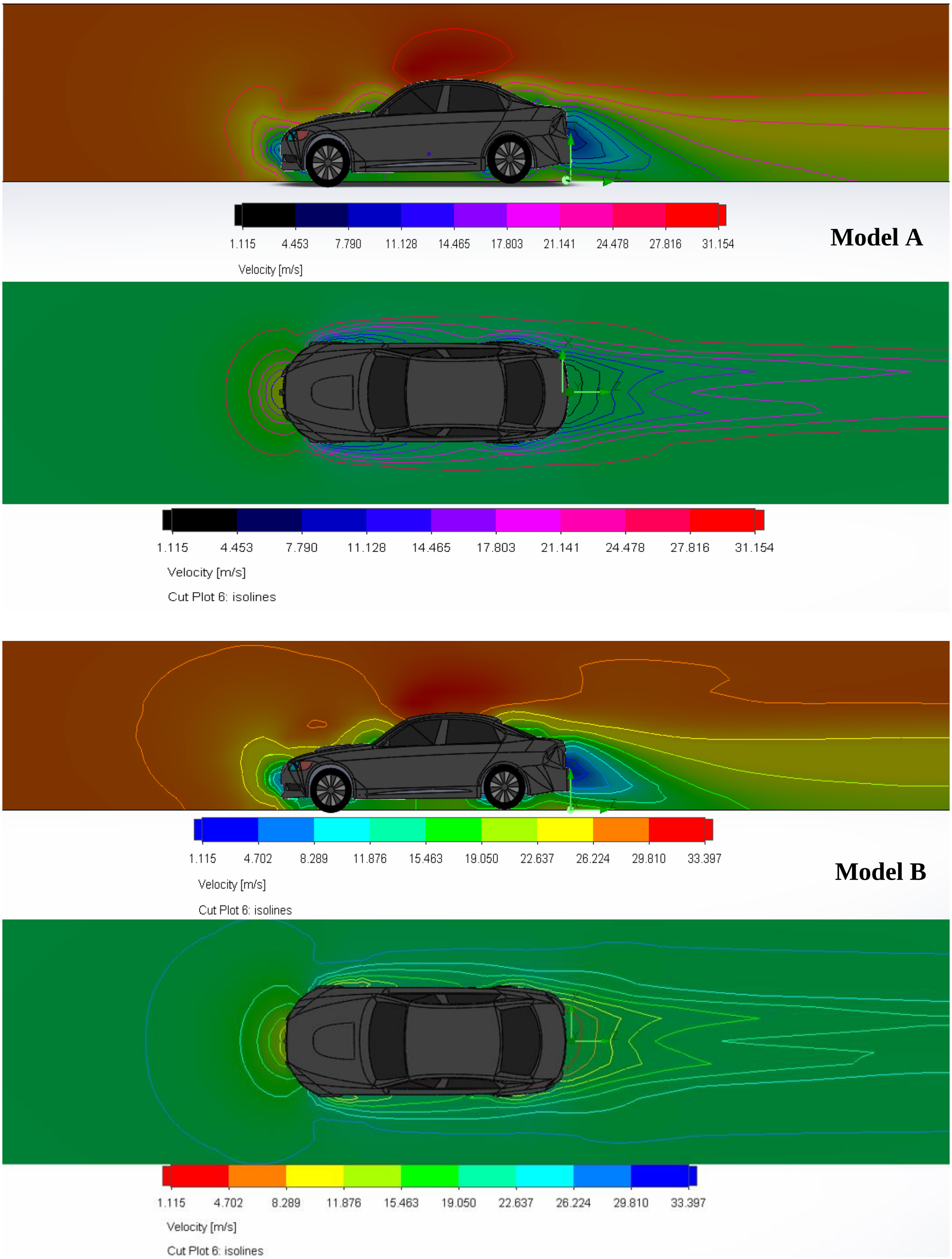


Figure 5. *Simulation results of velocity distribution around the vehicle of models A and B*

The air flow velocity after simulation is described in Fig. 5. It can be seen that at the front of the car, the two sides of the mirror and the end of the car, the velocity of the air flow is quite small because the air flow when going to this area will be stopped and separated to the two sides. Similar to simulate results the pressure distribution around the vehicle.

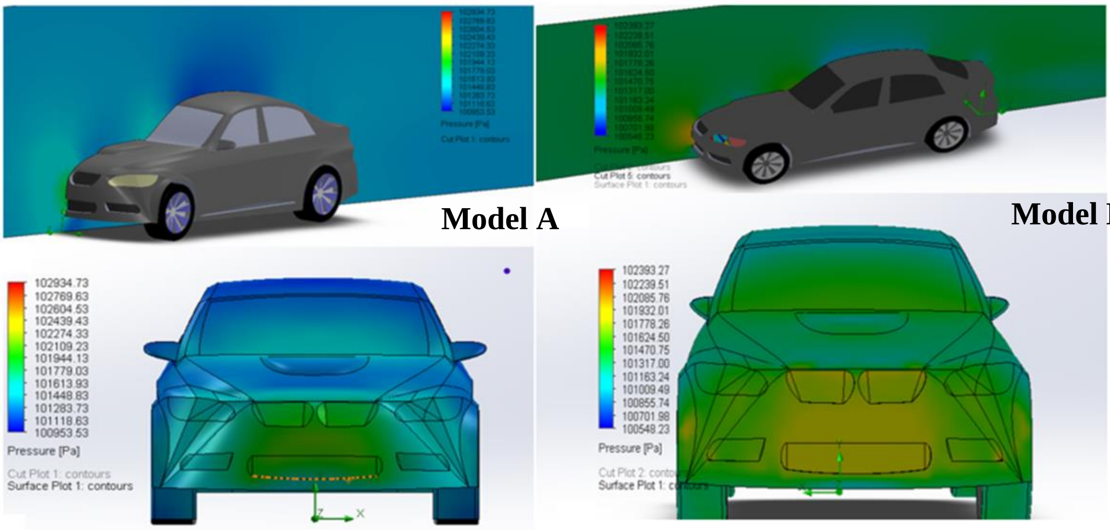


Figure 6. *Simulation results of pressure distribution around the vehicle of models A and B*

Above Fig. 6 shows the pressure acting on the surface of the vehicle in the vertical section and along the entire vehicle. The color bands from blue to red correspond to the pressure distribution at each point of the vehicle. It can be seen that the maximum pressure of the vehicle is at the front of the vehicle because it is directly affected by the wind in the perpendicular direction. The bonnet is the place where the least pressure is applied.

The final result is a simulation of the swirling air of the vehicle.

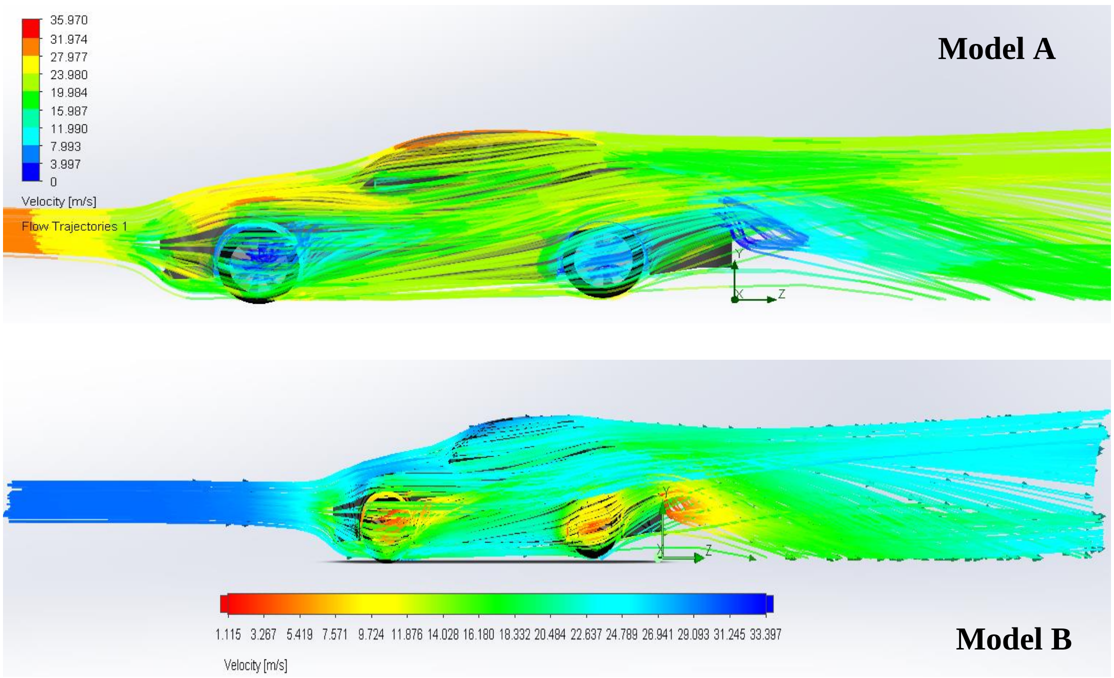


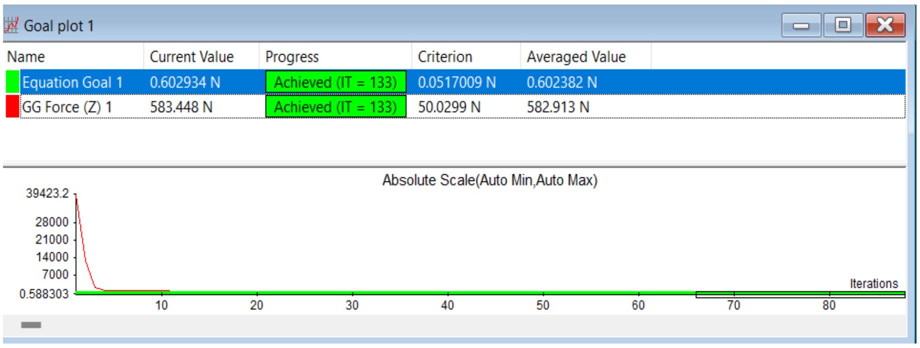
Figure 7. *Simulation results of the swirling air of models A and B*

Results in Fig. 7 show the distribution of vortex air flow before and after the improvement. After moving to the front of the car and separating to the sides, the air stream continues to go to the end of the car and create a tornado at a distance from the rear of the car. Observing both models, there are similarities in eddy currents and eddy current distribution. Eddy currents are the main cause of the pressure difference between the front and rear of the vehicle.

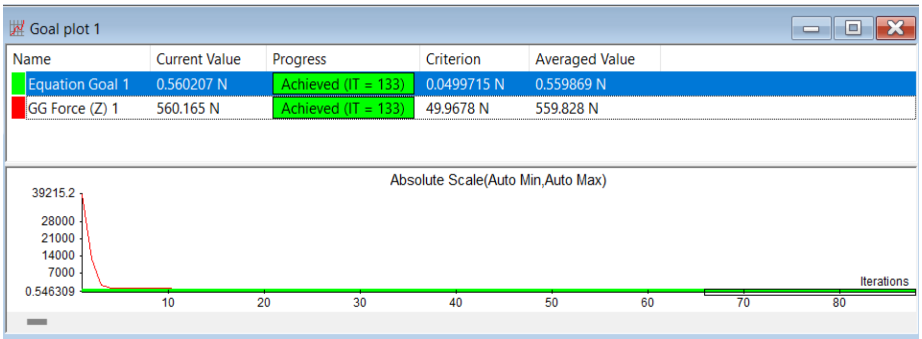
When the car is moving, the surface of the car is always subjected to a large impact force from air resistance (Loução et al., 2022), which is determined by formula (1):

$$Fd = 0.5 * \rho * Cd * A * U_{\infty}^{2} \qquad (1)$$

Where: Fd: aerodynamic drag (N)
ρ: density of air = 1.204 kg/m^3
Cd: aerodynamic drag coefficient
A: area of front bumper (m^2)
U_∞: speed of flow air (m/s)



Model A



Model B

Figure 8. *Calculation results of drag force and drag coefficient of two models*

The drag force Fd and drag coefficient Cd of the improved model are reduced compared to the original model. Specifically, drag Fd decreased from 582,913 N to 559,828 N and drag coefficient Cd decreased from 0.602382 to 0.559869. Because when implementing the design to change the mirror placement angle, we get a reduction in drag coefficient and drag compared to the original and improved efficiency.

4. Conclusion

The research uses Flow Simulation module in Solidworks to simulate and evaluate the results of side mirrors adjustment through values of pressure, velocity, and eddy current. After simulation and evaluation, drag reduced from 582,913 N to 559,828 N (4%), drag coefficient decreased from 0.602382 to 0.559869 (7.1%). Thereby we can see the importance of the side mirror in affecting the aerodynamic characteristics of the vehicle. This is an important premise to improve and design new style of sedan car that contribute to increasing the stability of today's cars.

References

Ayyagari, D. T. and He, Y. (2017). Aerodynamic analysis of an active rear split spoiler for improving lateral stability of high-speed vehicles. International Journal of Vehicle Systems Modelling and Testing, 12 (3/4):217. DOI:10.1504/IJVSMT.2017.089978.

Tsai, C. H., Fu, L. M.,Tai, C. H., Huang, Y. L. and Leong, J. C. (2009). Computational aero-acoustic analysis of a passenger car with a rear spoiler. Applied Mathematical Modelling 33, 9, 3661-3673. DOI:10.1016/j.apm.2008.12.004

Maji, D. S. B., & Mustaffa, N. (2021). CFD Analysis of Rear-Spoilers Effectiveness on Sedan Vehicle in Compliance with Malaysia National Speed Limit. Fuel, Mixture Formation and Combustion Process, 3, 1. https://fazpublishing.com/fmc/index.php/fmc/article/view/34

Hu, X. X. and Wong, T. T. (2011). A numerical study on rear-spoiler of passenger vehicle. World Academy of Science, Engineering and Technology, 57, 636-641. http://hdl.handle.net/10397/5189

Tunay, T., Yaniktepe, B. and Sahin, B. (2016). Computational and experimental investigations of the vortical flow structures in the near wake region downstream of the Ahmed vehicle model. Journal of Wind Engineering and Industrial Aerodynamics Volume 159, 48-64. DOI:10.1016/j.jweia.2016.10.006

Kanekar, S., Thakre, P. and Rajkumar, E. (2017). Aerodynamic study of state transport bus using computational fluid dynamics. IOP Conference Series: Materials Science and Engineering 263, 6, 062052. DOI:10.1088/1757-899X/263/6/062052

Muthuvel, A., Murthi, M. K., Sachin.N.P, Koshy, V. M., Shathi, S. and Selvakumar, E. (2013). Aerodynamic exterior body design of bus. International Journal of Scientific & Engineering Research, 4, 7, 2453-2457. https://www.academia.edu/29932774/Aerodynamic_Exterior_Body_Design_of_Bus

Abinesh, J. and Arunkumar, J. (2014). CFD analysis of aerodynamic drag reduction and improve fuel economy. International journal of mechanical engineering and robotics research, 3, 4, 430-440. http://www.ijmerr.com/show-129-574-1.html

Yadav, D., Chauhan, S., Karki, S., Nayak, S. R., Kumar, N. and Babu, S. S. (2017). CFD ANALYSIS FOR DRAG FORCE REDUCTION IN INTER-CITY BUSES. International Research Journal of Engineering and Technology (IRJET), 4, 5, 350-355. https://www.irjet.net/archives/V4/i5/IRJET-V4I567.pdf

Lakshmanan, B. and Yadav, S. K. (2020). Design And Analysis Of Rear Spoiler To Reduce Drag Coefficient Of A Sports Car. 8th International Conference on Contemporary Engineering and Technology 2020, Prince shri Venkateswara Padmavathy engineeering college, Chennai. https://www.researchgate.net/publication/343745005_Design_And_Analysis_Of_Rear_Spoiler_To_Reduce_Drag_Coefficient_Of_A_Sports_Car

Zaareer, M. and Mourad, A. (2022). Effect of Vehicle Side Mirror Base Position on Aerodynamic Forces and Acoustics. Alexandria Engineering Journal, 61, 2, 1437-1448. https://doi.org/10.1016/j.aej.2021.06.049.

Cheng, S. Y., Tsubokura, M. and Nakashima, T. (2012). Numerical quantification of aerodynamic damping on pitching of vehicle-inspired bluff body. Journal of Fluids and Structures, 30, 9, 188-204. DOI:10.1016/j.jfluidstructs.2012.01.002

Al-Obaidi, A. S. M. and Otten, W. A. (2018). Aerodynamic analysis of personal vehicle side mirror. Journal of Engineering Science and Technology, 13, 7, 52-64. https://www.researchgate.net/publication/326494098_Aerodynamic_analysis_of_personal_vehicle_side_mirror

Corallo, M., Sheridan, J. and Thompson, M. C. (2015). Effect of aspect ratio on the near-wake flow structureof an Ahmed body. Journal of Wind Engineering and Industrial Aerodynamics, 147, 95-103. https://doi.org/10.1016/j.jweia.2015.09.006

Tunay, T., Sahin, B. and Ozbolat, V. (2014). Effects of rear slant angles on the flow characteristics of Ahmed body. Experimental Thermal and Fluid Science, 57, 165–176. https://doi.org/10.1016/j.expthermflusci.2014.04.016

Dang, T. P., Zhengqi, G., Chenzhen, (2016). Numerical Simulation of the Flow Field around Generic Formula One. Journal of Applied Fluid Mechanics, 9, 1, 443-450. doi: 10.18869/acadpub.jafm.68.224.24260

Gukop, N. S., Kamtu, P., Lengs, B. D., Babawuya, A. (2021). Effect of Mesh Density on Finite Element Analysis Simulation of a Support Bracket. FUOYE Journal of Engineering and Technology, 6, 3. DOI:10.46792/fuoyejet.v6i3.632.

Loução, R., Duarte, G. O. and Mendes, M. J. G. C. (2022). Aerodynamic Study of a Drag Reduction System and Its Actuation System for a Formula Student Competition Car. Fluids, 7, 9, 309. DOI:10.3390/fluids7090309.