Parametric Investigation of Effect of Diffuser Angle on the Flow Characteristics of an Ahmed Body

Akash MB
M.Tech, Automotive Engineering
Vellore Institute of Technology
Vellore

Pranjal Patil
M.Tech, Automotive Engineering
Vellore Institute of Technology
Vellore

Tej Pratap Singh
M.Tech, Automotive Engineering
Vellore Institute of Technology
Vellore

***Abstract*—It is estimated that the aerodynamic drag is the governing form of resistance when vehicles run at speeds of 80 km/h or greater. Such high speeds compel the airflow to become turbulent and a low-pressure zone is created, inducing an uplift force and instability due to considerable amount of wake area generation. It is found that the installation of a diffuser with optimum angles can reduce the aerodynamic lift coefficient. In this research, aero-elastic analysis of rear diffuser on Ahmed body are investigated using numerical simulation technique in order to improve vehicle handling, stability and performance under high speed cornering maneuver situations .Initially, flow analysis is carried out on a simple Ahmed body and values of coefficient of discharge, lift and drag forces, pressure distributions are reported using commercially available simulation CFD software tool. Later, similar simulations are carried out by providing a diffuser angle on the Ahmed body and the resultant flow properties are compared with the previous ones. An appropriate diffuser angle is selected based on the derived results .The CFD-based simulation data thus obtained provides the basis for further optimization of the diffuser design for full scale high speed vehicle.**

Keywords— CFD,drag,Ahmed Body,Contours

I. INTRODUCTION

This An extensive research in the field of vehicle aerodynamics led to the improved performance and as well as increased the fuel efficiency of the modern vehicles. It is considered to be cost efficient and easier to modify the vehicle aero dynamics rather than trying to improve or tune for the efficiency of the power train which already reached a saturation state to achieve the same end results of increased speeds, reduced drag and increased downward forces [1]. The significant changes in the underbody structure and arrangement had benefitted the vehicle with lower drag and increased downward forces in particular with implementation of the diffuser. The function of an automotive diffuser is to accelerate the flow of air underneath the car, creating an area of low pressure, thereby increasing the down force for the vehicle to corner faster. The diffuser attempts to bring the low pressure air below the cars back to the ambient atmospheric pressure without inducing turbulence and filling the rear wake thus by reducing drag. The velocity of the air decreases as it moves along the diffuser.

Angel Huminic et al.[2] numerically investigated flow aspects in the rear portion of a standard Ahmed body with a diffuser and concentrated on the effect of variation of lift and drag on the aerodynamic characteristics for smaller slant angles. The authors did an extensive study on flow field around the vehicle and evaluated the values of critical slant angles above which airflow separation may occur. Pruthviraj Mohanrao Palaskar [3] discovered that side tapering of the diffuser structure tends to reduce the aerodynamic drag to a certain extent. Lasse Christoffersen et al.[4] systematically presented the interaction between the rear wheels, wings and the diffuser of a race car to achieve higher speeds and reduced drag . A depression at the diffuser inlet is presumed to be the chief source of down force. Moreover, the analysis dictates reduction in wake behind the car on increasing the diffuser angle.[5][6][7]

In the field of aerodynamics, the rear diffuser affects the rear flow field and wake patterns of vehicle significantly. Seung-On Kang et al. studied the aerodynamic drag reduction phenomenon employing an actively translating rear diffuser and observed a significant changes in the rear flow and wake patterns of the car leading to rise in pressure on the rear surface of a passenger car at high speed driving conditions normally above 70Km/h which effectively causes for the increased downward aerodynamic forces due to the reaction of the flow movement and as well reduced the drag. CFD simulations performed on the car-diffuser model with a variation in dimensions, configurations and driving speeds revealed that as the length of the diffuser increases, the drag reduction effect also increases [1]. In recent studies, drag coefficients for the Ahmed body was found out experimentally as a function of the yaw angle, ranging from 0° to 90°. F.J.Bello-Millán et al. conducted an experimental work in a wind tunnel test section to better understand the influence of yaw angle on coefficient of drag. [8] There is very little work published on newly designed diffusers with vanes that are seen on a lot of sports car. The vanes allow higher ramp angles and ride heights and work effectively when designed in combination with the diffuser. T. P. AniruddhanUnni carried out a research based on highly complex 3D flow around the vanes and flaps of a diffuser at various positions and angles to understand the flow features and arrive at an optimal solution to accomplish high vehicle performance with an adequate down force.[9] Apoorva Tyagi and N. Madhwesh worked on improving the performance of a under tray diffuser in terms of drag and down force by changing the geometric properties such as Inlet angle, Outlet angle analogous to varying ground clearances[10].

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 05, May-2018

From various slants angles of Ahmed body it is observed that flow separation starts after 25 degree of slant angle. Hence the simulation is done on ahmed body with 25 degree slant angle.

II. METHODOLOGY

The diffuser device blocks out the low-pressure air from the underbody and allows relatively high-pressure air from the side and the upper surface to fill the space at the rear surface of the ahmed body, which results in an increase in the base pressure. According to the configuration of the device, it generates a diffuser effect which increases the underbody flow pressure, thus increasing the base pressure of the car body.

The main purpose of this study is to explain the aerodynamic drag reduction mechanism of an ahmed body by introduction of a diffuser angle on the rear surface. The base pressure of the rear surface is increased as it prevents the low pressure air coming through the underbody from directly soaring up to the rear surface . At the same time, a diffusing effect occurs in the underbody which lowers the velocity and raises the pressure, bringing about an aerodynamic drag reduction.

By keeping the diffuser length constant and varying the diffuser angles, we are analyzing 4 different diffuser configurations. All the above cases are tested at a velocity of 90 km/hr (25 m/s). After creating a physical computational domain around the using "ANSYS DESIGN MODELLER, we are importing it in to "ANSYS WORKBENCH" meshing tool and surface is first covered with all quad elements with hexa dominant mesh extending the entire domain.

Test section is simulated in "ANSYS FLUENT" for a series of calculations and the results are presented in order to show the effects of velocity and pressure distribution along with the prediction of coefficient of drag and lift at different locations.

2.1. Geometry of Ahmed body

An Ahmed body is the component under research here .Fig.1 shows the complete CAD representation of it.

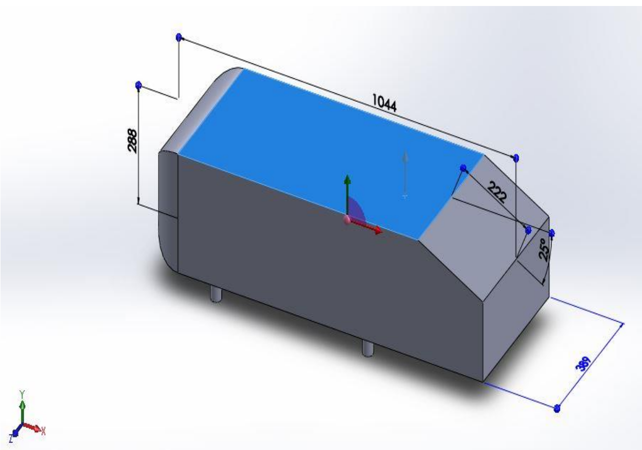


Figure 2.1. CAD model of Ahmed Body

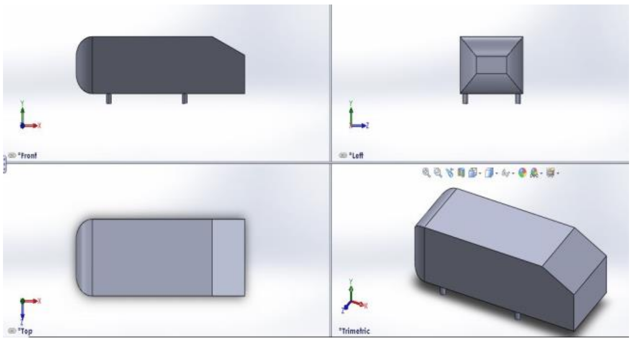
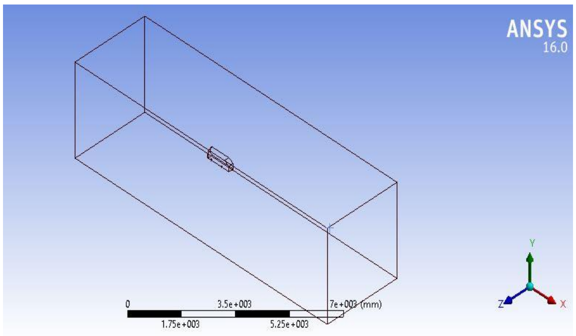


Figure 1.Different views of Ahmed Body



2.2.Enclosure Generation:

Figure 2.2. Enclosure

For the numerical studies an idealized computational domain with a constant rectangular cross-section was used. As shown in Figure 3, the computational domain's L × W × H is 11544mm × 3338mm × 3194.5mm. The domain extended around seven times the vehicle length to the front and three times to the rear.

2.3.Mesh Generation:

Mesh Type	Hex Dominant
Inflation	Program controlled
Growth rate of meshing	1.2 %

Table 2.1 Mesh Details

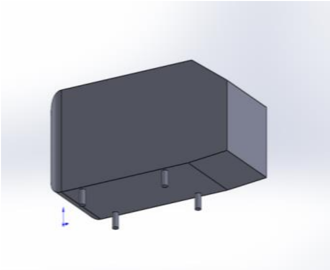
For the same degree of polynomial, the finite element space generated by hexahedral elements is more rich than the space generated by tetrahedral elements. Hence, Hexa dominant meshed component is used for analysis. A moderate mesh was preferred to get good results.

Statistics	
Nodes	317298
Elements	1436539
Mesh Metric	None

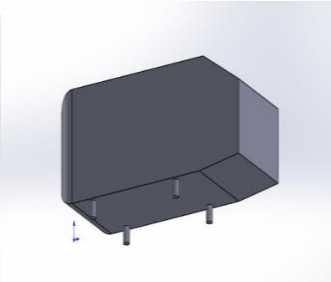
Table 2.2.Mesh statistics

2.4. Modelling of Ahmed body with various diffuser angles

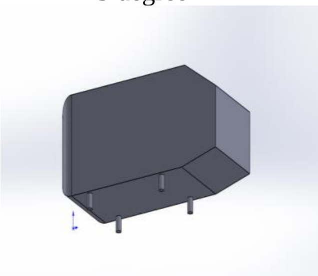
Five different cases including original model were designed. The diffuser angle was set to 0°, 3°, 6°, 9° and 12° respectively for each case while the diffuser angle of the original model is 0°. Length of the diffuser is constant i.e. 222 mm that has been taken based on the research work undertaken till date for analysis. The isometric views for ahmed body with diffuser for different cases were shown in Fig 2.4.



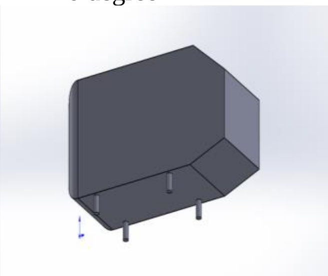
3 degree



6 degree



9 degree



12 degree

Figure.2.4. Ahmed body configurations

These 3D models will be simulated under same conditions as that of the Ahmed body to get drag and lift forces and coefficients.

III. RESULTS AND DISCUSSION

The important parameters considered for any CFD simulation are geometry creation and its integration in a physical domain, grid generation and choice of turbulence model and computing scheme. The numerical simulation in this work was done in the commercial code FLUENT. Due to its stability and ease of convergence ,the k-epsilon model was the chosen turbulence model. Realizable k-epsilon model gives the best match with the experimental results over other models and is chosen in the vicinity of walls as it provides superior performance for flows involving rotation, separation and recirculation, boundary layers under adverse pressure gradients. Both first order upwind discretization schemes and second order upwind discretization schemes were used for the momentum, turbulent kinetic energy and turbulent dissipation. Coupled based scheme was set as the iterative algorithm; the residual value was set to 0.0001. Turbulent intensity at inlet and outlet boundaries is set as 0.8 for first order upwind discretization and 0.95 for second order upwind discretization based on the research carried out so far in simulation of Ahmed body.

Inlet boundary condition (velocity)	25 m/s
Outlet boundary condition (Gauge pressure)	0 Pascal
Wall zones	No slip
Yaw angle	0°

Table 3.1. Boundary Conditions

Solver used	Pressure based
Solution method	Coupled
Turbulence model	k-Ɛ (2 equations) , realizable, Non-equilibrium conditions

Table 3.2.Solution Details

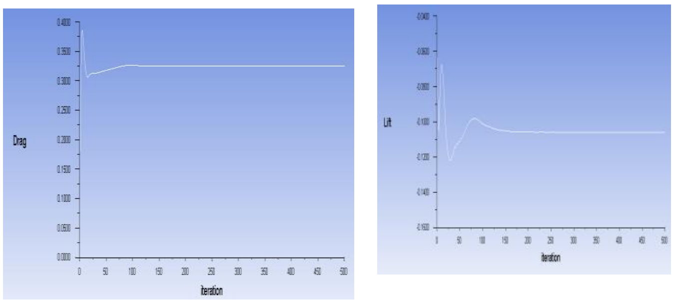


Table 3.1.Convergence plots for drag and lift coefficients

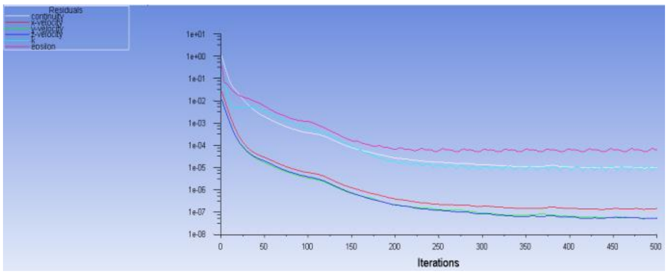


Figure 3.2. Scaled Residuals Plot

Generally, a scaling factor is employed in the simulation process to cut short the number of iterations and save computational time. But it leads to reduction in the accuracy of results and erroneous results. Hence, to get accurate results, a minimum scaling factor and maximum number of iterations i.e. 500 have been considered for simulation purpose.

For Ahmed body:

Serial no.	Slant angle(in degrees)	Coefficient of drag	Coefficient of drag
1	25	0.379	0.330
2	30	0.383	0.4222

Table 3.3 Cd and Cl results for various slant angles

For Ahmed Body with diffuser:

Serial no.	Diffuser angle(in degrees)	Coefficient of drag	Coefficient of lift
1	3	0.3481	0.2428
2	6	0.354	0.1098
3	9	0.339	0.0138
4	12	0.325	-1.059

Table 3.4. Cd and Cl results for various slant angles

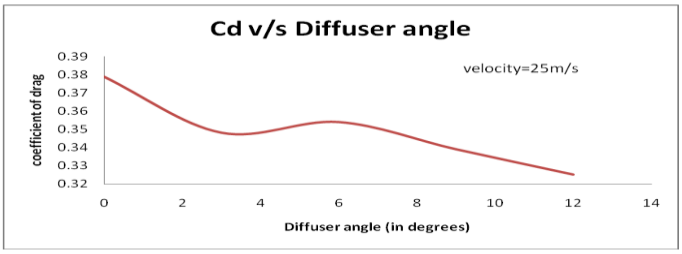


Figure3.3. Diffuser angle (degree) vs. Coefficient of drag

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 05, May-2018

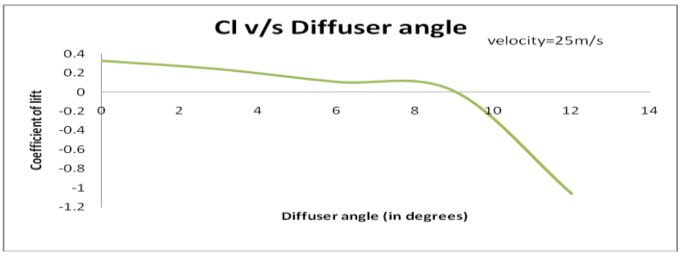


Figure3.4.. Diffuser angle (degree) vs. Coefficient of lift

Total drag and lift coefficient for various diffuser angle of the Ahmed body was shown in Table 3.4. Fig 3.3. and Fig 3.4. shows the curve of total drag and lift coefficient versus diffuser angle. We observe that when the diffuser angle varied from 0°to 12°, the total aerodynamic drag coefficients of Ahmed body first decrease and then increase slightly and decrease further with increasing diffuser angle , while the total aerodynamic lift coefficients go on decreasing. There is a diffuser angle at which the Ahmed body can obtain the minimum drag coefficient i.e. 12° in this particular case.

With the increase of diffuser angle, the distribution area of positive pressure on the rear of the body first increases and then decreases. Difference of positive pressure distribution on the rear of the body lead to differential pressure of the body surface that varies from case to case, which results in the total aerodynamic drag coefficients of body first decreasing and then increasing while diffuser angle changes.

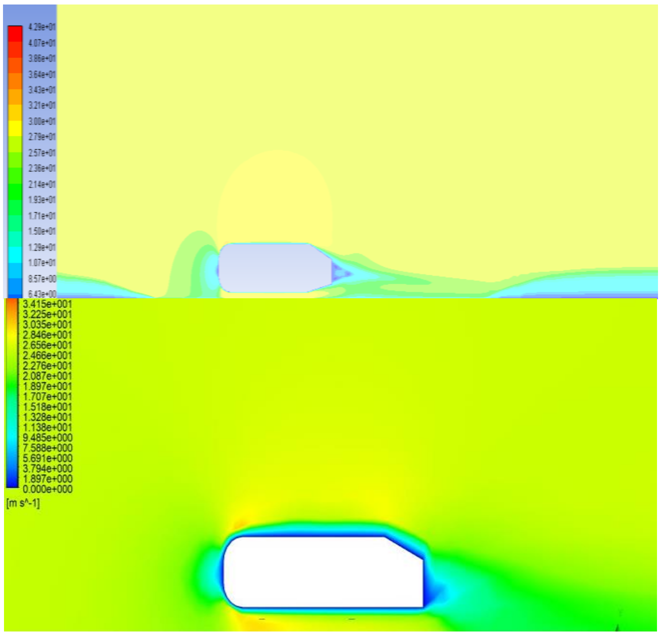


Figure.3.5.Velocity contour for Ahmed body diffuser(12°) and with diffuser

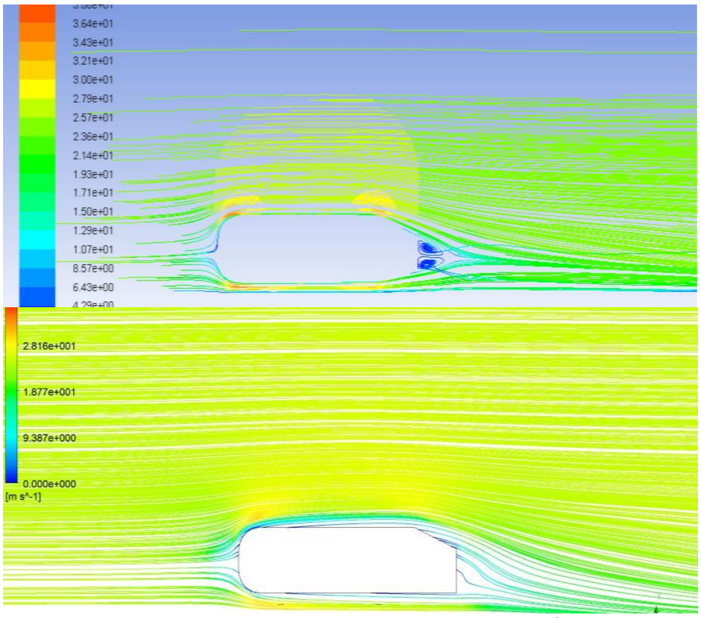


Figure 3.6. Velocity Pathlines for Ahmed body with diffuser(12°) and without diffuser

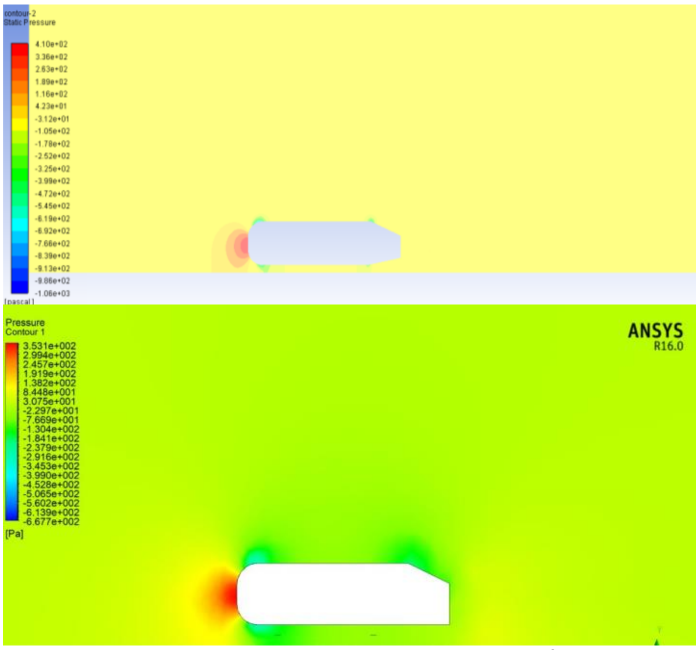


Figure **3.7**. Pressure contour for Ahmed body with diffuser (12°) and without diffuser

The pressure contours of the Ahmed body with diffuser and simple Ahmed body about the symmetric wall zone are displayed to compare each other. As confirmed from the pressure color, the base pressure of Ahmed body with diffuser angle is increased by the diffuser device and it pushes the body forward. It decreases the aerodynamic drag of the Ahmed Body. As the diffusion effect occurs, the streamline patterns change rapidly and that varied streamline forms different pressure contour result in base pressure increment. Eventually, the flow from the underbody is controlled and compressed air by the diffusing process which is the core reason for the aerodynamic drag reduction phenomenon as depicted in Figure 3.7.

IV. CONCLUSION

- Airflow over the Ahmed body is investigated using the commercial code Fluent CFD software to understand the flow processes involved in drag production. The configurations included different rear slant angles and are compared with the experimental results of Ahmed model. It is found that the maximum drag for an Ahmed body is with rear slant angle of 30^{o}.
- Later Aerodynamic lift, drag and flow characteristics of the ahmed body with different diffuser angles are numerically investigated. In order to induce negative lift in the ahmed body, we need to install diffusers which will create a down-force by creating a negative pressure on bottom side and positive on top side and thereby providing more stability. Installation of diffuser will also restricts the flow separation and boundary layer formation thereby reducing the drag force which is resisting the motion of the body
- The analysis was performed considering the half scale model for ease of analysis. Therefore, the diffuser performance needs to be checked with the full scale of ahmed body configuration for accurate results.
- Finally, optimization method of the diffuser shape is considered as the appropriate future work in this area.

REFERENCES

[1] Kang, S., Cho, J., Jun, S., Park, H. et al., "A Study of an Active Rear Diffuser Device for Aerodynamic Drag Reduction of Automobiles," SAE Technical Paper 2012-01-0173, 2012, https://doi.org/10.4271/2012-01-0173.
[2] Huminic, A. and Huminic, G., "Computational Study of Flow in the Underbody Diffuser for a Simplified Car Model," SAE Technical Paper 2010-01-0119, 2010, https://doi.org/10.4271/2010-01-0119.
[3] Palaskar, P., "Effect of Side Taper on Aerodynamics Drag of a Simple Body Shape with Diffuser and without Diffuser," SAE Technical Paper 2016-01-1621, 2016, https://doi.org/10.4271/2016-01-1621.
[4] Christoffersen, L., Söderblom, D., Lofdahl, L., Hill, R. et al., "Wing-Diffuser Interaction on a Sports Car," SAE Technical Paper 2011-01-1433, 2011, https://doi.org/10.4271/2011-01-1433.
[5] Unni, T., "Numerical Investigation on Aerodynamic Effects of Vanes and Flaps on Automotive Underbody Diffusers," SAE Technical Paper 2017-01-2163, 2017, https://doi.org/10.4271/2017-01-2163.
[6] Hu, Xingjun, et al. "Influence of different diffuser angle on Sedan's aerodynamic characteristics." Physics Procedia 22 (2011): 239-245
[7] Huminic, A. and Huminic, G., "Numerical Flow Simulation for a Generic Vehicle Body on Wheels with Variable Underbody Diffuser," SAE Technical Paper 2012-01-0172, 2012, https://doi.org/10.4271/2012-01-0172.
[8] Marklund, J., Lofdahl, L., Danielsson, H., and Olsson, G., "Performance of an Automotive Under-Body Diffuser Applied to a Sedan and a Wagon Vehicle," SAE Int. J. Passeng. Cars - Mech. Syst. 6(1):293-307, 2013, https://doi.org/10.4271/2013-01-0952.
[9] Keating, A., Shock, R., and Chen, H., "Lattice Boltzmann Simulations of the Unsteady Flow Behind the Ahmed Body," SAE Technical Paper 2008-01-0740, 2008, https://doi.org/10.4271/2008-01-0740.
[10] Kohri, I., Yamanashi, T., Nasu, T., Hashizume, Y. et al., "Study on the Transient Behaviour of the Vortex Structure behind Ahmed Body," SAE Int. J. Passeng. Cars - Mech. Syst. 7(2):586-602, 2014, https://doi.org/10.4271/2014-01-0597.