Original Research Article

The Numerical Investigation of Aerodynamic Structures of Truck and Trailer Combinations

C. Bayindirli[1*], Y. E. Akansu[2], M. S. Salman[3], D. Colak[4]

[1] Niğde University, Nigde Vocational School of Technical Sciences, Turkey
[2] Niğde University, Faculty of Engineering, Mechanical Engineering Department, Turkey
[3] Gazi University, Faculty of Technology, Automotive Engineering Department, Turkey
[4] Poligon Engineering Product Development and Manufacturing Technologies, Turkey

Abstract

In this study, aerodynamic structures of the heavy vehicle consisting of truck and trailer was investigated with computational fluid mechanics method. The force measurement were performed on the model car and aerodynamic drag coefficient (C_D) were determined numerically in 4 different speeds on the Fluent® program. Numerical analysis of flow were made on the 59 000 - 844 000 Reynolds number. The effect of trailer additional to truck was determined to aerodynamic drag coefficient. The friction and pressure induced distributions were determined of total aerodynamic resistance. The images of flow structure were obtained around the truck trailer. The zones that was forming aerodynamic resistance were determined.

Key words: Aerodynamic drag coefficient (C_D), turbulent flow, truck and trailer, computational fluid mechanics

*Corresponding Author:
E-mail: cbayindirli@nigde.edu.tr

1. Introduction

The aerodynamics is called as a science that investigates the interaction of moving of solid bodies with the air. The aerodynamics vehicles significantly affect the car's performance, fuel consumption, acceleration properties, handling characteristics, environmental pollution, noise and comfort [1, 2, 3, 4, 5]. Moreover, the cooling system and the heating of interior have a direct relationship with the aerodynamics. Aerodynamic drag coefficient increases proportionally with the square of the speed. [6, 7, 8] This status makes the improving aerodynamics resistance more important issue for heavy vehicles which perform a large part of the transportation out of the city and a lot of miles at high speeds a year. A passenger car with 100 km speed an hour spends 60% of its power to defeat the forces of drag resistance. The car spends 20 %of its power to defeat wheels friction force and the rest of power to overcome other forces that prevent the movement. Considerable savings is achieved from the fuel consumption with the improvement of the aerodynamic properties of the vehicle [9, 10, 11, 13 and 14]. Perzon and Davidson [12] conducted a study on a heavy vehicle to improve aerodynamic properties of it. By rounding the back of the trailer he achieved 4 % improvement. He stated that he made 3% improvements with nose cone and 7% improvements with chassis skirt [12].

The aim of this study is to examine the aerodynamics structures of truck and trailer combinations and determinate of zones where is forming aerodynamic resistance as numerically. To view and investigate the turbulent flow around the truck and trailer by CFD methods.

2. Numerical Analysis Method

In this study, aerodynamics structures of the 1/32 scale truck trailer combination was investigated by computational methods in fluid mechanics (CFD). In CFD analysis as a model car a truck and standard trailer have chosen. The truck is producing and most common finding in Turkey. Prototype truck trailer and 1/32 scale model car are given in Fig.1 and Fig.2.

Fig. 1. Prototip vehicle

Fig.2. 1/32 scaled vehicle

2.1. Obtaining Of Drawing Datas

In vehicle aerodynamics studies ensuring the geometric similarity is one of the fundamental similarity conditions. In this study three-dimensional drawing data is obtained by three-dimensional optical scanning device with the using optical scanning method. Three-dimensional optical scanning device is given at Fig.3.

Fig.3. Three-dimensional optical scanning device

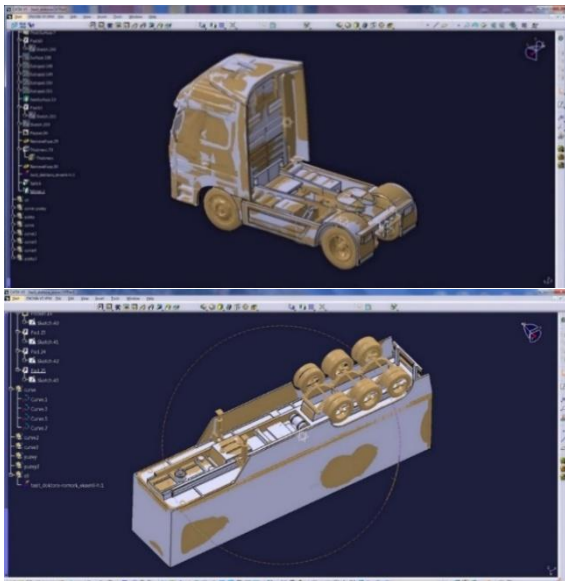Fig.4. Three-dimensional scan of model car

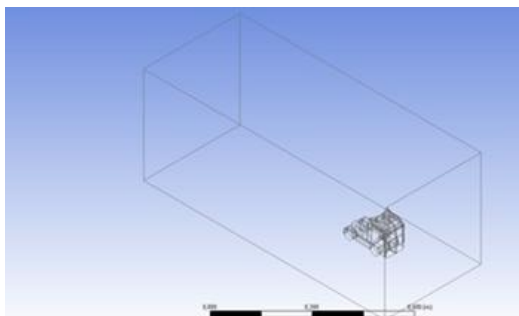
Fig.5. Drawing datas of truck and trailer

Fig.6. The mold cavity of truck

Blockage ratio is defined as projection area of the front surface of the model, proportional to area of the front surface of wind tunnel test section. In the literature, blockage ratio is recommended to be below the 10 % limit for the blocking effect to be neglected in wind tunnel tests [2]. In this study, blocking ratio is 8.31% for truck and trailer combination. As this value is in accordance with the criteria given in the literature, the effects of blockade have been neglected.

The truck and trailer mold cavity is formed at Fig.6 and Fig.7. Test area measurements is taken as 40 cm x 40 cm x 100 cm in the CFD analysis.
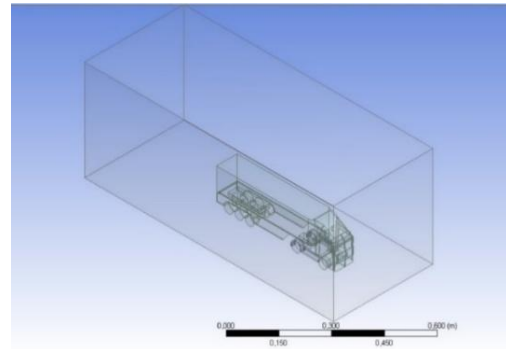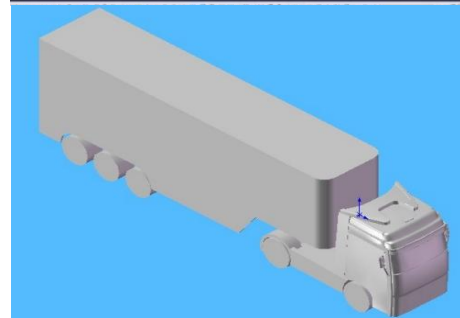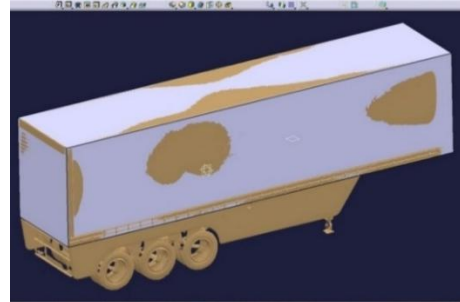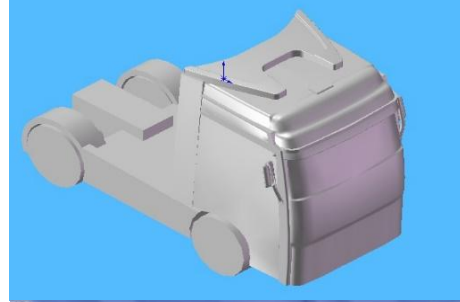

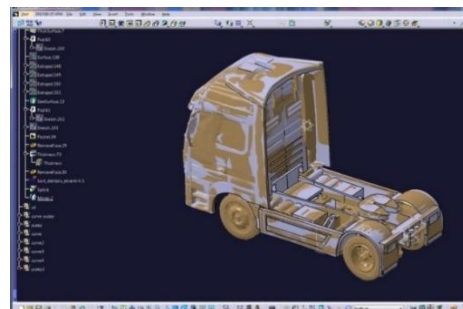Fig.7. The mold cavity of truck and trailer

Fig.8. Simplified drawing datas

2.2. The assumptions made in drawing data in numerical analysis

In the meshing the complexity of the drawing data, small details of drawing and layers give the mesh error that is smaller than the minimum mesh size and numerical analysis cannot be done.

Thus, it is seen in Fig. 8 small pieces closed, without complex and simpler components has been added to model car which are disrupt the flow structure and geometric similarity. The errors has been neglected resulting from this situation in the numerical analysis results.

3. CFD Analysis

Numerical flow analysis was carried out in Fluent® program and computers with features of Intel® CoreTM i5 3570 CPU, 3.40 GHz processor and 8 GB ram. The analyzes are made in. 10m / s, 15m / s, 20m / s and 27 m / s.

3.1. Design modeler

It is seen in Fig.9. The drawing datas of truck and trailer are defined to program in this section and transferred to the meshing section.
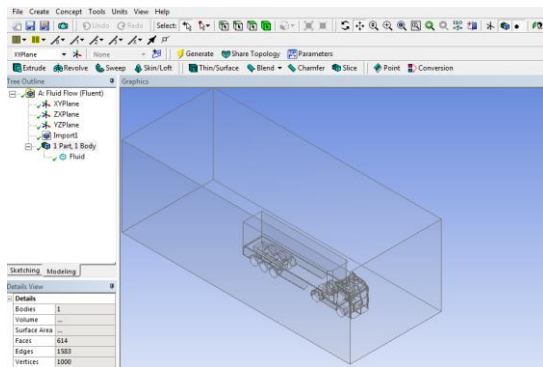

Fig.9. The interface view of truck and trailer combination in design modeler section

3.2. Meshing

1 071 921 triangular volumes cell structures (tedrahedrons) are formed for truck and 1 681 381 for truck trailer combination. The analysis domain boundaries are defined as follows.

✓ **Inlet:** The surface of entry of the fluid is defined as a condition of constant speed limit.

✓ **Outlet**: The surface of the fluid exit is defined as a condition of constant pressure boundary.

✓ **Wall:** The wall boundary condition is used.

The mold cavity of truck and trailer combination: The wall boundary condition is used.
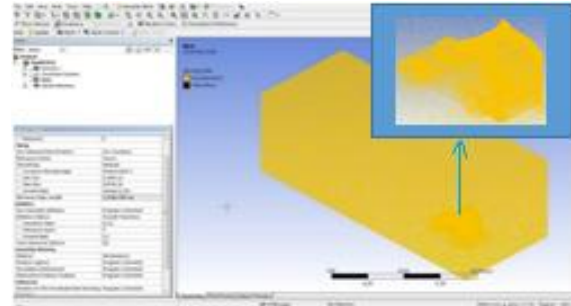

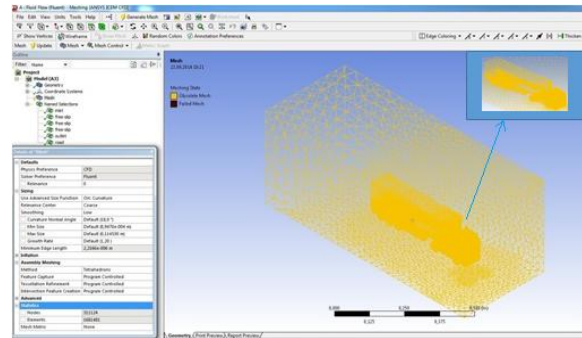Fig.10. The interface view of truck in meshing section

Fig.11. The interface view of truck and trailer combination in meshing section
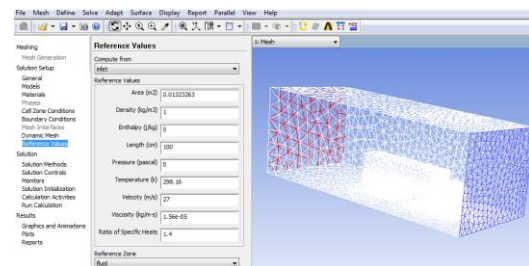

Fig.12. The interface view of truck and trailer combination in setup section

3.3. Setup

In this study, the convergence criteria is taken as 1.0×10^{-3} for continuity, x-velocity y-velocity and z-velocity. Turbulence intensity is taken as 1%. Front projection area of the truck is calculated as 0.0108 m^2 and 0.0132 m^2 for truck trailer combination from reports-projected area in Fluent® program. Analyzes was carried out using SIMPLE Cell Based Least Squared, RNG k-ε turbulence model,

standard wall functions and standard initialization. The boundary conditions is used which are given Table 1 – 4.
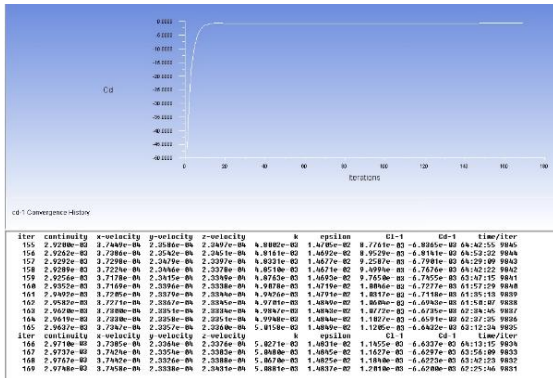

Fig.13. The graph drag force coefficient of truck and convergence

Table 1. The used features in the numerical analysis

Function	Feature
Solver	Segregated
Formulation	Implicit
Time	Constant
Speed formulation	Absolute
Change options	Node-based
Fluid	Incompressible air
Pressure Speed connection	Simple

Table 2. Relaksasyon criterias

Feature	Value
Pressure	0.3 Pa
Density	1 kg/m^3
Body forces	1N
Momentum	0.7 m^2/s
Turbulence kinetic energy	0.8 m^2/s^2
Turbulence dissipation rate	0.8
Turbulent viscosity	1 kg/m.s

Table 3. Air properties

Feature	Definition	Value
ρ	Density	1 kg/m^3
μ	Dynamic viscosity	1.560×10^{-5} kg/m.s

Table 4. Interpolation options

Feature	Value
Pressure	Second-Order
Momentum	Second-Order
Turbulence kinetic energy	First-Order
Turbulence dissipation rate	First-Order

Aerodynamic drag coefficient is expressed with the parameters of drag force FD, the density of air ρ, the free stream velocity V and as the front projection area of vehicle A.

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A} \qquad (1)$$

After the numerical analysis drag force and aerodynamic resistance coefficient was calculated. Flow lines have been obtained in the form of color scale vector and streamline Furthermore, the pressure distribution images were obtained on the truck.

4. Result
4.1. Numerical Analysis Results of Truck

According to numerical results the average aerodynamic resistance coefficient (C$_D$) is calculated 0.649 at the 10 m/s, 15m/s, 20m/s and 27 m/s speeds. 98.4% pressure induced and 1.6% friction induced of the total aerodynamic drag coefficient were found.

Table 5. Numerical Analysis Results of Truck

Flow rate (m/s)	Force (N)	C$_D$
10	0,343	0,633
15	0,817	0,670
20	1,420	0,655
27	2,524	0,638

The shape of front geometry and attack angle of windshield affects the pressure induced aerodynamic resistance.
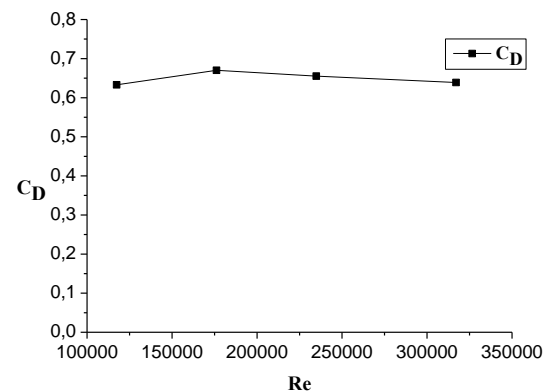

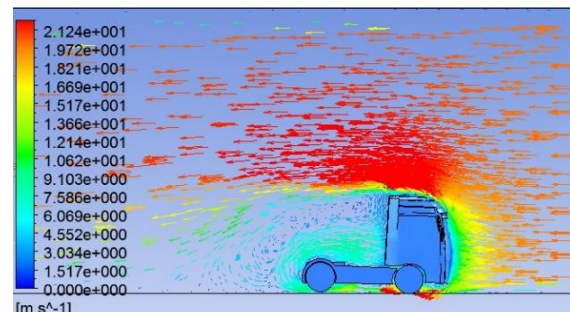Fig.14. Aerodynamic drag coefficient (C$_D$) of truck

Fig.15. The vector image of the wind speed at Re=235000

It is seen in Fig.17 due to the small attack angle of windscreen the pressure coefficients was observed high on the vehicle's front glass and of the front bumper.
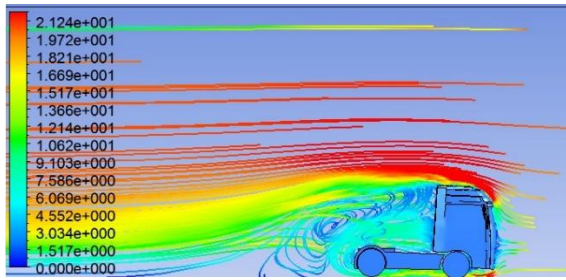
143

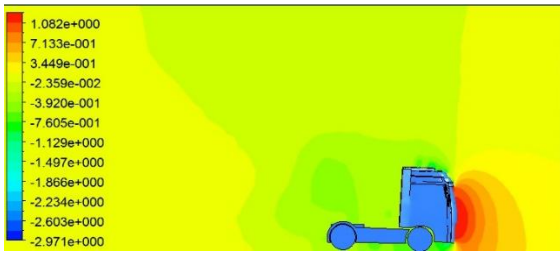Fig.16. The streamline image of the wind speed at Re=235 000

Fig.17.The pressure coefficient (C_P) distribution on the truck at Re=235 000

4.2. Numerical Analysis Results Of Truck and Trailer Combination

According to numerical results the average aerodynamic resistance coefficient (C_D) is calculated 0.776 at the 10 m/s, 15 m/s, 20 m/s and 27 m/s speeds.

It was found that 94.2% pressure induced, 5.8% friction induced of the total aerodynamic drag coefficient. In this study, the attached trailer to tractor has increased the aerodynamic resistance coefficient C_D 19.59%.

Table 6. Numerical Analysis Results of Truck and Trailer

Flow rate (m/s)	Force (N)	C_D
10	0,524	0,792
15	1,111	0,746
20	2,075	0,784
27	3,775	0,782

It is seen in Fig.19 and Fig.20 the flow speed is too low on front bumper and front glass. Besides a turbulence is formed rear of trailer. The size of the turbulent region commonly affect to total aerodynamic resistance. It was observed that air flow between the truck and trailer. The air speed is very slow on trailer wheels.

As seen in Fig.21, the pressure coefficient very high on front bumper and windscreen. Besides the pressure coefficient is high part of the trailer higher truck.
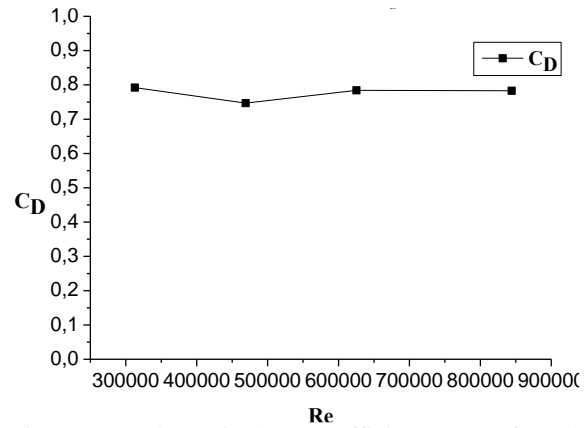

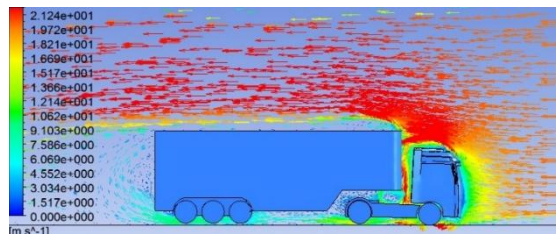Fig.18. Aerodynamic drag coefficient (C_D) of truck and trailer combination

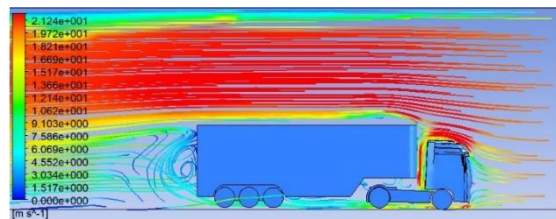
Fig.19. The vector image of the wind speed at Re=625000

Fig.20. The streamline image of the wind speed at Re=625 000

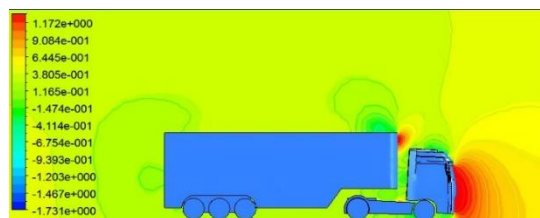Fig.21.The pressure coefficient (C_P) distribution on the truck and trailer combination at Re=625 000

5. Conclusion and Recommendations

In this study the flow structure on the tractor trailer model of, the forces acting on the vehicle values were determined numerically. The numerical flow analysis is made in the range of 59000-844000 Reynolds numbers. The effect of added the trailer to aerodynamic drag coefficient was determined. According to numerical results the average aerodynamic resistance coefficient (C_D) is calculated 0.649 of truck at the 10 m/s, 15 m/s, 20 m/s and 27 m/s speeds. It was found that 98.4% pressure

induced, 1.6% friction induced of the total aerodynamic drag coefficient. The aerodynamic resistance coefficient (C_D) is calculated 0.776 of truck and trailer at the same speeds. It was found that 94.2% pressure induced, 5.8% friction induced of the total aerodynamic drag coefficient. The attached trailer to tractor has increased the aerodynamic resistance coefficient C_D 19.59%. It was observed in the flow of images that the pressure coefficients is very high on the front glass. The windshield angle of attack should be more narrowly in order to reduce aerodynamic drag. It has been shown that aerodynamic improving obtainable with accordance designed spoiler.At streamline image of flow the air speed is very slow on trailer wheels. The aerodynamic improvements can be obtained by closing trailer wheels.

8. References

1. Çakmak, M.A," Investigation of vehicles as aerodynamically", Mühendis Makina, 41, 489, 2000.
2. Wahba, C. E.M., Al-Marzooqi, H., Shaath, M., Shahin, M., and El-Dhmashawy. T. (2012). Aerodynamic Drag Reduction for Ground Vehicles using Lateral Guide Vanes. CFD Letters Vol. 4(2), 68-78.
3. Gilhaus, A, "The influence of cab shabe on air drag of trucks", Journal of Wind Engineering and Industrial Aerodynamics, 9, 77-87, 1981.
4. Chowdhury, H, Moria H, Ali A, Khan I, Alam, F, Watkins, S, "A study on aerodynamic drag of a semi-trailer truck" 5th BSME International Conference on Thermal Engineering Volume 56, Pages 201–205, 2013.
5. Miralbes, R, "Analysis of Some Aerodynamic Improvements for Semi-Trailer Tankers" Proceedings of the World Congress on Engineering Vol 3 WCE, 4-6 July, London U.K, 2012.
6. Çengel, A, Y, Cimbala J, M, "Fluid Mechanics Fundamentals and Applications", (Çev. Tahsin Engin, H. Rıdvan Öz, Hasan Küçük, Şevki Çeşmeci), Güven Bilimsel, İzmir, 2008.
7. Sahin C, "Prediction of Aerodynamic Drag Coefficient for Heavy Vehicles with Computational Fluid Dynamics Method" İstanbul Technical Universty, Institute of Science and Technology, Master Thesis, 2008.
8. Apisakkul, K.T., and Kittichaikarn, C. (2005). Numerical analysis of flow over car spoiler. Paper presented The Ninth Annual National Symposium on Computational Science and Engineering Papers ANSCSE-9, Bangkok, Thailand.
9. Desai M., Channiwala S. A., Nagarsheth, H. J. (2008). Experimental and Computational Aerodynamic Investigations of a Car. WSEAS Transactions on Fluid Mechanics Issue 4, Volume 3, 359-366.
10. Hu, Xu-xia., and Wong, E.T.T. (2011). A Numerical Study On Rear-spoiler Of Passenger Vehicle. World Academy of Science, Engineering and Technology, 57, 636-641.
11. Lokhande, B., Sovani, S., and Khalighi, B. (2003). Transient simulation of the flow field around a generic pickup truck. SAE Technical Paper Series, 01-1313, 1- 19.
12. Perzon, S., and Davidson, L. (2000). On transient modeling of the flow around vehicles using the Reynolds equation. International Conference on Applied Computational Fluid Dynamics (ACFD) Beijing China, 720-727.
13. Vaghela, K. (2013).Optimization of Roof Fairing Angle to Reduce the Aerodynamic Drag of Heavy Duty Truck. International Journal of Emerging Technologies in Computational and Applied Sciences IJETCAS, 13-322, 113-117.
14. Içingür, Y., Solmaz, H. (2011). Determination of Drag Coefficinets of Various Automobile Models in a Low Speed Wind Tunnel. Journal of the Faculty of Engineering and Architecture of Gazi University, 2(455-460).