International Journal of Mathematics and Physics 6, №1, 8 (2015)

UDC 519.63; 519.684

*Issakhov A., Khan Y., Temirbekuly N.

Faculty of Mechanics and Mathematics, al-Farabi Kazakh National University,
Almaty, Kazakhstan
*e-mail: alibek.issakhov@gmail.com

Numerical modelling of detached flow around a car body by using large eddy simulation method

Abstract. This paper presents a mathematical model of detached flow around a car body by using large eddy simulation method, which is solved by the equations of Navier - Stokes for an incompressible fluid based on the method of splitting by physical parameters that can be discretize by the control volume method. In the first step it is assumed that the transfer of momentum carried out only by convection and diffusion. Intermediate velocity field is solved by 5-step Runge - Kutta method. At the second stage, the pressure field is solved based on the found intermediate velocity field. The algorithm is parallelized on high-performance systems. The obtained numerical results of three-dimensional detached flow around a car bodyreveals to approximate qualitatively and quantitatively the basic laws of hydrodynamics processes occurring in aerodynamics.

Key words: Ahmed body, Navier-Stokes equations, finite volume method, Runge-Kutta method, pressure drag.

Introduction

As increasing the number of producing an automotive becomes evident automotive industry require more fuel efficiency as nowadays the problem is important for ecology and the costs. Most fuel efficiency depends on drag and to generate fuel efficient vehicle one must pay attention to drag reduction. As it is known the drag generally is made by pressure drag that generates at rear end [2].

Ahmed benchmark model is the model that predicts most importance characteristics of the flow over a bluff body [1]. The Ahmed reference model is a car-like bluff body, representing a highly simplified ¼ scale lower medium size hatchback vehicle with a slant back. Besides relatively simple geometry, the flow around Ahmed body retains some main features of the flow around real cars. The model's major attributes are 1044mm x 389mm x 288mm. The flow regimes are fully turbulent as the Reynolds number based on body length is usually too high. The flow around ground vehicle contains several separation regions.

Many experimental and numerical works have made in these area upon Ahmed body model and real cars. Numerical research can predict some aerodynamic characteristics and minimize costs of experimental work. As experimental research providing qualitative and quantitative understanding numerical research is also important for interpreting the experiments and to obtain more complete understanding in flows around bluff body.

From Ahmed experimental measurements in a wind tunnel [1] that most suitable for vehicle testing it was revealed that 85 percent of drag comes from pressure drag or form drag [1], generating at a rear end and detailed analysis were made upon the time averaged wake structure. The wake flow behind the vehicle is the flow region which represents major contribution to car's drag. For Ahmed used model based length Reynolds number was 4.29 million. It was only slant angle of changed geometric parameters. There are other experimental works with use of Ahmed body reference model such as [3, 7-9].

Chenguang Lai et al. [4] were performed both experimental and numerical investigation to analyze the influence of the different rear diffuser angle on aerodynamic drag and wake structure. Also experimental and numerical investigation with use of CFD models were made by Ivan Korkischko et al. [6]

for better understanding and validation between experiments and adequate numerical simulation.

Large eddy simulation was performed by Prasanjit Das et al. [5] with a slant back angle of 25. Some numerical investigation involve RANS based models [10-15] and LES based models [16-20] to predict main features of the flow around ground vehicles.

Mathematical model

Therefore, flow around vehicle can be described by the equations in the Boussinesq approximation. For mathematical modeling of the system of equations are considered, which including the equations of motion and the continuity equation. The development of three-dimensional turbulent flow around vehicleis considered. Three-dimensional mathematical model is used for modeling of flow around vehicle [23, 24]:

$$\frac{\partial \overline{u_i}}{\partial t}+\frac{\partial \overline{u_j}\,\overline{u_i}}{\partial x_j}=-\frac{\partial \overline{p}}{\partial x_i}+\nu\frac{\partial}{\partial x_j}\left(\frac{\partial \overline{u_i}}{\partial x_j}\right)-\frac{\partial \tau_{ij}}{\partial x_j}, \quad (1)$$

$$\frac{\partial \overline{u_j}}{\partial x_j}=0, \quad (i=1,2,3), \qquad (2)$$

where $\tau_{ij}=\overline{u_i u_j}-\overline{u_i}\,\overline{u_j}$, g_i – gravitational acceleration, β – expansioncoefficient, u_i – velocity components.

Smagorinsky turbulence model is used to close the system of equations (1) - (3) [21].

The finite volume method is used for the numerical simulation. For this purpose let us represent the Navier-Stokes equation and equation for the temperature in the form of integral conservation laws for arbitrary fixed volume Ω with boundary $d\ \Omega$ [23, 24]:

$$\int_\Omega\left(\frac{\partial U}{\partial t}+\frac{\partial F_i}{\partial x_i}+\frac{\partial G_i}{\partial x_i}-B_i\right)d\ \Omega=0, \qquad (4)$$

where

$$U=\begin{pmatrix}0\\ u_j\\ \end{pmatrix},\ F_i=\begin{pmatrix}u_i\\ u_i u_j+p\delta_{ij}-\tau_{ij}\\ \end{pmatrix},$$

$$G_i=\begin{pmatrix}0\\ \nu\frac{\partial \overline{u_i}}{\partial x_j}\\ \end{pmatrix},\ B=\begin{pmatrix}0\\0\\ \end{pmatrix}.$$

equation (4) can be written in this form

$$\int_\Omega\left(\frac{\partial U}{\partial t}-B\right)d\ \Omega+\oint_{\partial\Omega}(F_i+G_i)\ n_i d\ \Gamma=0. \qquad (5)$$

And we can write the equations (5) like

$$\int_\Omega\left(\frac{\partial U}{\partial t}\right)d\ \Omega+\oint_{\partial\Omega}(F_i+G_i)\ n_i d\ \Gamma=\int_\Omega B_i d\ \Omega. \qquad (6)$$

Grid functions are defined at the cell center, and fluxes across the border in divided cells. Cell volume is denoted by the grid functions.

Now we perform a discretization of equation (6) by the control volume (CV) and control surface (CS)

$$\sum_{CV}\left(\frac{\Delta U}{\Delta t}\right)\Delta\Omega+\sum_{CS}(F_i+G_i)n_i\Delta\Gamma=\overline{B_i}\Delta\Omega \qquad (7)$$

or we can write the equation (7) in this form:

$$\sum_{CV}\Delta U\Delta\Omega+\sum_{CS}\Delta t(F_i+G_i)n_i\Delta\Gamma=\Delta t\overline{B_i}\Delta\Omega. \qquad (8)$$

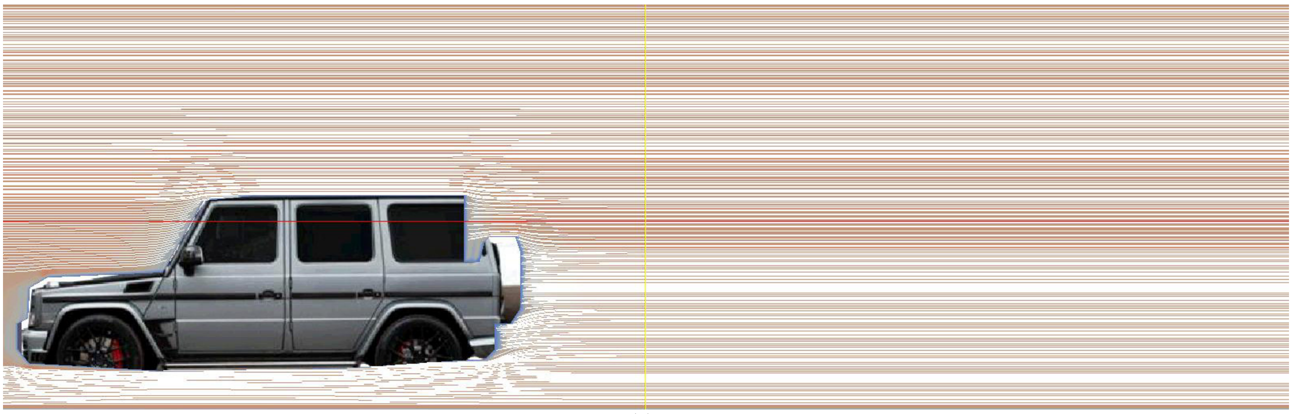
Numerical algorithm

Splitting method by physical parameters is used to solve the equation (1) – (2) [22, 23, 25]. Discretization in form of (8) is used for the numerical implementation of the system (1) – (2). In the first step it is assumed that the transfer of momentum carried only by convection and diffusion. The intermediate velocity field is solved by 5-step Runge - Kutta method. In the second stage, the pressure field is found based on the found intermediate velocity field. Poisson equation for the pressure field is solved by Jacobi method. The third step it is assumed that only the transfer is carried out by pressure gradient.The algorithm is parallelized on high-performance systems. Simulations were made on cluster systems URSA and Cluster of Institute of Mathematics and Mechanics at the al-Farabi Kazakh National University.

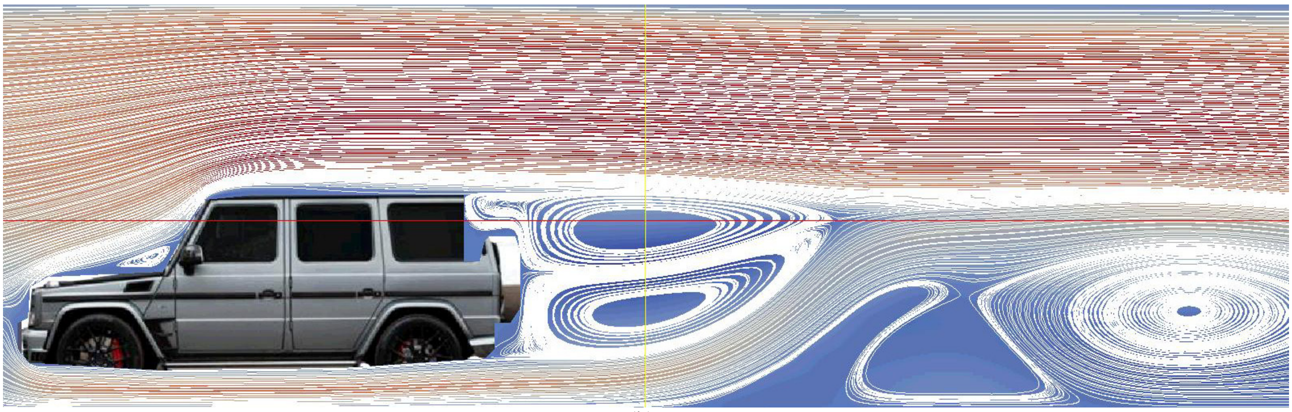
$$\text{I)}\ \int_{\Omega} \frac{\vec{u}^{*}-\vec{u}^{n}}{\tau}\, d\,\Omega = -\oint_{\partial\,\Omega} \left(\nabla \vec{u}^{n}\, \vec{u}^{*} - \nu \Delta \vec{u}^{*} \right) n_i d\,\Gamma,$$

$$\text{II)}\ \oint_{\partial\,\Omega} \left(\Delta p \right) d\Gamma = \int_{\Omega} \frac{\nabla \vec{u}^{*}}{\tau}\, d\,\Omega,$$

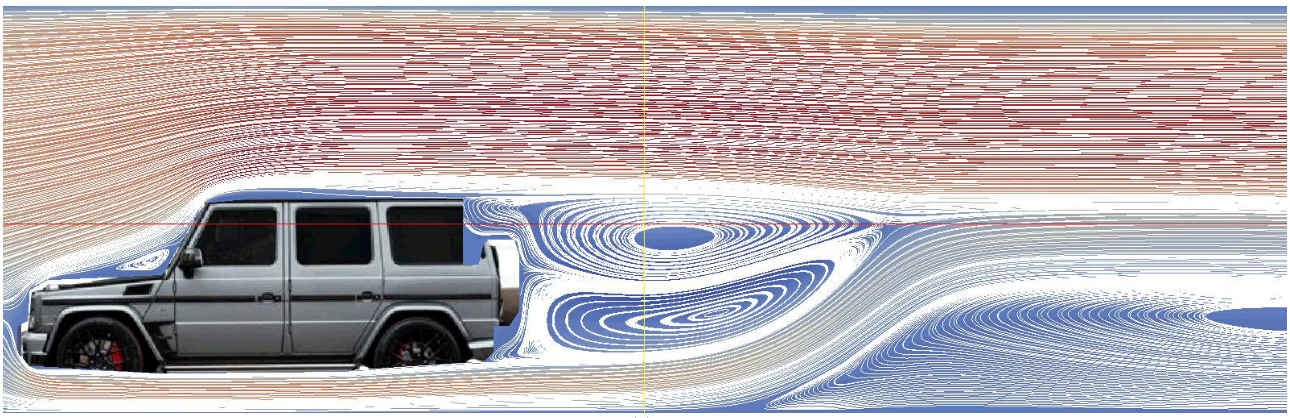
$$\text{III)}\ \frac{\vec{u}^{n+1}-\vec{u}^{*}}{\tau} = -\nabla p,$$



(a)

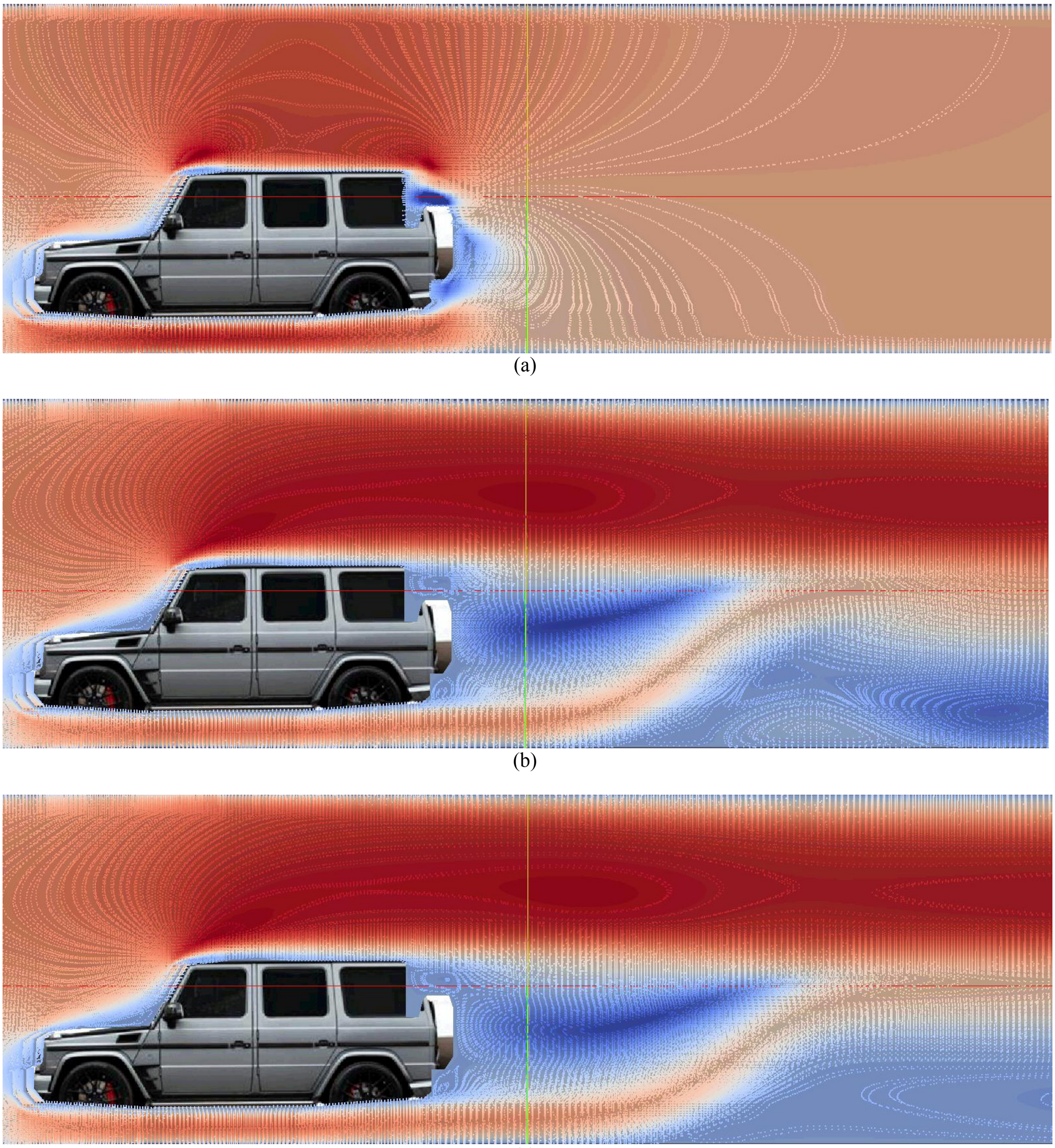


(b)



(c)

Figure 1 - Flow visualization with velocity profiles and streamlines at time: (a) t=0; (b) t=0.5; (c) t=1



(a)

(b)

(c)

Figure 2 - Flow visualization with velocity profiles and particles at time: (a) t=0; (b) t=0.5; (c) t=1

Results of numerical simulation

Initial and boundary conditions were determined to solve the problems. More than 800 000 grid points were used in simulations. Further it's presented numerical results obtained for Mercedes Gelandewagen car. Numerical results were obtained with LES technique using Smagorinsky SGS model. As it's shown from figures 1 and 2 for this type of car with the slant angle of 0 degree we can see a small vortex in front of the car at low part of windshield that was discussed in M. Minguez et al. [19]. In this work it's shown the flow separates and then reattaches on the leading edge and at sharp edge between the roof of Ahmed body and the slant surface which slant was 25 degree at z=0 plane. Also it was observed that the flow around Ahmed body was turbulent and completely three dimensional with differences going from center to sides of the given model. The results presented in the given work are two dimensional with planes for the future of three dimensional one. As observed in experimental [3] and numerical works [19] the flow separates at the beginning of the slant and then reattaches in the middle of the slant. In M. Minguez et al. [19] work the flow separates at the two edges between the slant and the lateral surfaces of the Ahmed body. In the present study with given 0 degree of the slant angle Figures 1 and 2 show that the flow separates at the roof and the bottom of the rear end that lead to two large counter-rotating trailing vortices with time going by spreading father in the wake.

References

1. Ahmed S.R., Ramm G., Faltin G. Some salient features of the time-averaged ground vehicle wake. SAE Paper 840300, – 1984.
2. Happian-Smith J. Introduction to modern vehicle design. Transport research laboratory (TRL), UK.
3. Lienhart H., Becker S. Flow and turbulence structure in the wake of a simplified car model. SAE Paper 2003-01-0656. – 2003.
4. Lai Ch., Kohama Y., Obayashi Sh., Jeong Sh.. Experimental and numerical investigations on the influence of vehicle rear diffuser angle on aerodynamic drag and wake structure. International Journal of Automotive Engineering 2, – 2011. – P. 47-53.
5. Das P., Sayem A.S.M., Islam M.T. Large-Eddy Simulation of the flow around an Ahmed reference model. Asian congress of Fluid Mechanics, 17-21 December, Dhaka, Bangladesh.
6. Korkischko I., Romano Meneghini J. Experimental investigation and numerical simulation of the flow around an automotive model: Ahmed Body. 19th International Congress of Mechanical Engineering, Proceedings of COBEM2007, Brasilia.
7. Bayraktar D., Landman D., Baysal O. Experimental and computational investigation of Ahmed body for ground vehicle aerodynamics, SAE Report, 01-2742, – 2001.
8. Spohn A., Gillieron P. Flow separations generated by a simplified geometry of an automotive vehicle, IUTAM Symposium: unsteady separated flows. Toulouse, France, – 2002.
9. Sims-Williams D. Experimental investigation into unsteadiness and instability in passenger car aerodynamics, SAE Report, 980391, – 1998.
10. Manceau R., Bonnet J.P. Workshop on refined turbulence modelling. 10th ERCOFTAC _SIG-15_/IAHR/QNET-FD, – 2002.
11. Gillieron P., Chometon F. Modelling of stationary three-dimensional detached airflows around an Ahmed reference body. Third International Workshop on Vortex, ESAIM Proceedings 7, – 1999. – P. 173-183.
12. Durand L., Kuntz M., Menter F. Validation of CFX-5 for the Ahmed car body. CFX Validation Report No. CFX-Val 13/1002, – 2002.
13. Han T. Computational analysis of three-dimensional turbulent flow around a bluff body in ground proximity. AIAA J. 27, –1989. – P. 1213-1219.
14. Menter F.R., Kuntz M. Development and application of a zonal DES turbulence model for CFX-5. CFX internal report, – 2003.
15. Guilmineau E. Computational study of flow around a simplified car body. J. Wind. Eng. Ind. Aerodyn. 96. – 2007. –P. 1207-1217.
16. Krajnovic S., Davidson L. Flow around a simplified car. J. Fluids Eng. 127, 907 _2005_; 127, – 2005. – P. 919–928.
17. Howard R.J.A., Pourquie M. Large eddy simulation of an Ahmed reference model. J. Turbul. 3(1), – 2002, art no. 012.
18. Hinterberger M., Garcia-Villalba M., Rodi W.The aerodynamics of heavy vehicles: Trucks, Buses, and Trains. Applied and Computational Mechanics, New York, 2004.
19. Minguez M., Pasquetti R., Serre E.High-order large-eddy simulation of flow over the “Ahmed body” car model. Physics of fluids, 20(9), 095101, – 2008.
20. Franck G., Nigro N., Storti M., D’elía J. Numerical simulation of the flow around the Ahmed vehicle model. Latin American Applied Research 39. – 2009. – P. 295-306.
21. Lesieur M., Metais O., Comte P. Large eddy simulation of turbulence. New York, Cambridge University Press, 2005. P.219.
22. Issakhov A. Large eddy simulation of turbulent mixing by using 3D decomposition method. Issue 4 (2011) J. Phys.: Conf. Ser.318. PP. 1282-1288, 042051. doi:10.1088/1742-6596/318/4/042051
23. Chung T. J. Computational Fluid Dynamics. Cambridge University Press, 2002 – P. 1012.
24. Fletcher C.A. Computational Techniques for Fluid Dynamics. Vol 2: Special Techniques for Differential Flow Categories, Berlin: Springer-Verlag. -1988. – P. 485.
25. Issakhov A. Pryamoe chislennoe modelirovanie (DNS) turbulentnih techenii s ispolzovaniem parallelnih tehnologi. Bulletin of KazNU, -2012. – № 2(73). – P.81-91