Advances in Dynamical Systems and Applications.
ISSN 0973-5321, Volume 16, Number 2, (2021) pp. 1547-1556
© Research India Publications
https://www.ripublication.com/adsa.htm

Eddy Viscosity Turbulent Flow Simulation over a Simplified Ahmed Body Model

Sujit Mishra, Ashok Misra*

Centurion University of Technology and Management, Odisha, India.

**Corresponding Author. Ashok Misra*

Abstract

The purpose of this study is to simulate turbulent flow over an Ahmed body in an open test wind tunnel. Two types of meshing, namely tetra prisms (TP) and hex prisms (HP), are employed to solve the partial differential equations associated with two equation turbulence models, namely Realizable k-ε (RKE) and Shear stress transport k-ω (SST k-ω), which are analysed over the 25^{o} slant Ahmed body. The drag coefficient, velocity profiles, and turbulent wake intensities are all numerically solved and illustrated in figures and tables. It is observed that the SST k-ω, turbulence scheme with hex-prism mesh produces results that are comparable to those of existing schemes and hence can be employed for aerodynamic simulation research.

Keywords: Ahmed model, URANS turbulence models, Turbulent flow, Aerodynamics

1. INTRODUCTION

In earlier years, investigation of external aerodynamics with ground vehicles has been mostly done with experimental and analytical techniques. With the advancement in time, and keeping the view of energy conservations, the efficiency enhancement has become crucial for ground vehicles such as a passenger car. At present, computational fluid dynamics (CFD) governing with Navier-Stokes equations gains popularity and used as the tool for analysing the external flow phenomena with the power and availability of computers successfully. This complements the solutions such that it could even replace the experimental measurement setups. Traditionally, the setup involves discretization of the computational domain into a small group of elements called building blocks that can take shape as structured and unstructured grids. This enables to apply over various numerical schemes at different boundary conditions. Many codes have been written and still in a developing stage to establish the proper estimation of aerodynamic parameters. This is because every CFD code has its own features that can be applied over various real time problems. Prior knowledge is highly essential to know its specific roles. Moreover, researchers always tried to gain results at an optimum grid with less computational time, which is only possible with improvising the numerical schemes.

Studies have been carried out in these areas, which helps to decide the accuracy and scalability of the relevant numerical schemes used for the aerodynamic parameters. (Le Good and Garry, 2004)has reviewed the designs of various reference scaled models used in the production's development process of vehicles in automotive aerodynamics fields. But the most well-known universal model develop (Ahmed SR. et al., 1984) known as Ahmed model has been focused in most of the studies because of its simplified geometry that helps researchers to communicate their findings and together make further advancements. The results investigating through experimental technique's for the flow profiles over the slant surface to estimate the drag coefficient which has contributed the whole car industry till date as the argument of reference. (Fares, 2006) has predicted the capability and feasibility in lattice Boltzmann solver taking a very large eddy simulation (VLES) framework over the 25^{o}slant Ahmed model but found difficulties in attaining the time steps that required for the flow parameters towards converging.(Guilmineau, 2008) has proposed a CFD simulation technique called as a quadratic explicit algebraic stress model (EASM) that has been applied to the Ahmed body with 25^{o}and 35^{o}slant angles. In both cases, the scheme could predict 3% less values of the drag than that of the other scheme results. Further, it has been found that all the simulations scheme used in the study with a massive flow separation particularly at 25^{o}slant body posing a strong challenge towards the establishing of turbulence schemes.(Morgut and Nobile, 2012) has done and emphasis on the influence of two turbulence schemes such as Shear Stress Transport (SST) and Baseline-Reynolds Stress Model (BSL) used over hexa- structured meshes with hybrid- unstructured meshes around the marine propellers. There is a slightly better prediction of flow fields in the BSL scheme than the SST turbulence model.(Serre et al., 2013) has investigated two different eddy- resolving modelling approaches, i.e. Large Eddy Simulations (LES) and Detached Eddy Simulation (DES) schemes around the 25^{o}slant Ahmed body. In their work, they reveal out the issues arises for the computational cost and ease implementation in their schemes without effecting the quality of results. Similar studies from (Aljure, Lehmkuhl, Rodríguez, and Oliva, 2014) has been presented a comparative study about the flows and turbulent structures through LES modelling with two various types of simplified models. (Mishra et al., 2017; Tientcheu-Nsiewe et al., 2016) investigates the Reynolds Averaged Navier–Stokes (RANS) equations over the Ahmed model taking second order upwind scheme to get the results of different flow parameters viz. drag force, drag coefficient, turbulent kinetic energy, and wake flow structures. Moreover, using a hybrid RANS-LES model from (Ashton et al., 2016)benefits than the RANS models in terms of the force coefficients, and general flow field for the Ahmed car body. But being more expensive than RANS, they further suggested improving RANS modelling keeping in view of potential cost benefits. (Moghimi and Rafee, 2018) has studied that focused on optimising the diffuser angle on the Ahmed body using SST k-ωturbulence scheme. They found that the performance in their applied scheme had been the best choice in the estimation of aerodynamic parameters. Multiphase simulation approaches open up new dimensions to investigate ground vehicles aerodynamics. Recent work by(Gaylard et al.,2017) through a very large eddy simulation (VLES) scheme has shown the water sprayed particle effects and its deposition pattern in the wakes region. Also a similar study, from (Mishra et al. 2019) that contains air and sand particles flows over the 2D Ahmed model used for the investigation of the aerodynamic flow parameters. In their investigation, un-steady RANS (URANS) mixture scheme shows a rise in the in drag coefficient, turbulent kinetic energy which has occurred because of high sand volume fraction levels.

Overall above investigations, it is well understood that, in order to provide effective simulation results, the methodology utilized should be independent of CAD geometry, computational domain topological dimensions, meshing strategy, and physical modelling. A working knowledge of physics, its geometric complexity, and processing capacity is always suggested and required. Thus, the current work makes an attempt by applying Eddy viscosity turbulence schemes, namely the RKE and SST k-models, to the hexa-prism (HP) and tetra-prism (TP)mesh in open wind tunnel with Ahmed Body to extract the necessary unsteady time-average and large scale flow quantities at a reasonable computational cost.

2. COMPUTATIONAL DOMAIN AND BOUNDARY CONDITIONS

The computational domain specification for the generic Ahmed model is chosen from previous work (Guilmineau, 2014) such that the aerodynamic forces are not affected due to blockage effects. The geometry size of the Ahmed model is 1044mm in length, 389 mm in width, and 288 mm in height, respectively shown in the 1. The slant rear part is 222mm long, making a 25^{o} with the base. The model without legs is kept inside the computational domain at 50mm distance from the ground road surface. Half model due to symmetry is chosen for the present analysis has it enhances the efficiency of the CFD process.

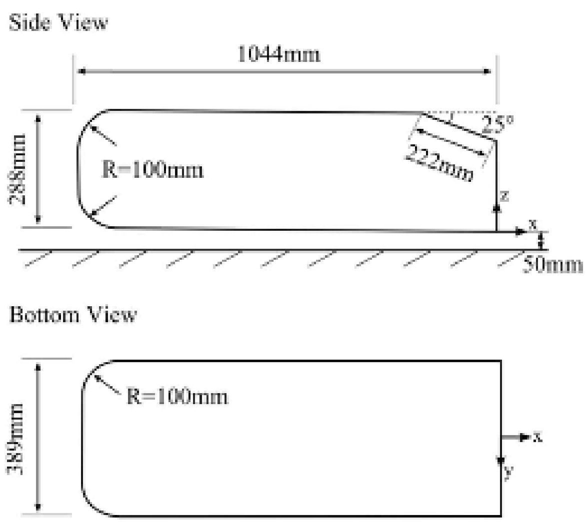


Figure 1. Geometrical Specifications of Ahmed Model

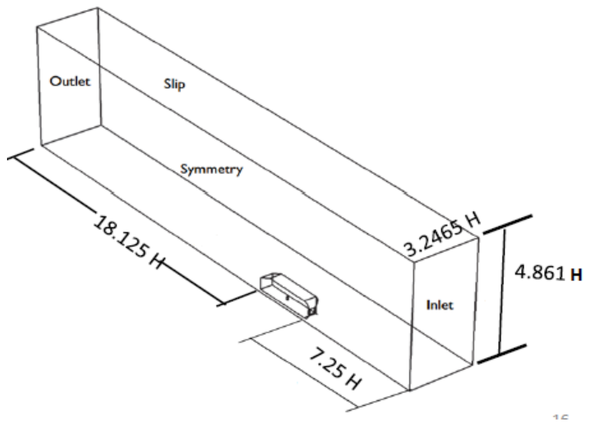


Figure 2. Computational Domain of Model

The computational domain in the present study consists of surface and volume cells created for two types of meshes, such as unstructured and structured grids as shown in figure (3) and (4). Further to capture all details of the flow phenomenon, for both the types of meshes the first cell height distance from the surface body is set equal to y+ of 10 with the appropriate wall function. A total of 8 prism layers with agradually increasing rate of 1.2 and the thickness of the boundary layer are calculated to be 0.0251m. The incoming free stream velocity flow at the inlet was set to 40m/s and the Reynolds number based on height remain same as in the experiments (Guilmineau et al., 2018) conducted in the open test section with a blockage ratio of 4%. At the outlet boundary are set to zero pressure conditions. The rest of the parts of the computational model are set to stationary walls with no-slip condition, as shown in Figure (2).The flow simulation at the incidence angle 25^{o}Ahmed model produces a strong 3-dimensional turbulent wake, implying that the turbulent intensity is high. The values for the turbulent intensity are taken as 1% based on the experiments (Guilmineau 2008; Guilmineau et al., 2018).

3. GRID INDEPENDENCE

The purpose of using both grid systems is to establish the influence on the drag coefficient. Thus, the meshes were refined further at the rear part in particular to capture the clear structure of the wakes by using unsteady flow RANS models.

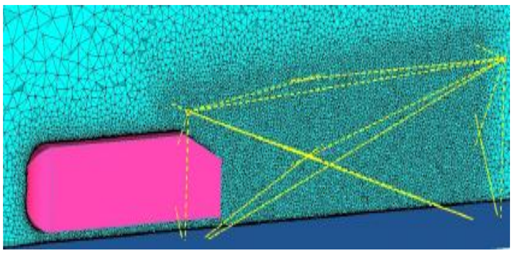


Figure 3. Tetra-prism Unstructured Mesh

For each type of grid structure, 3 types of meshing, such as coarse, medium, and fine, are used to perform an independence test and to find an appropriate grid level. Unsteady RKE modellingisperformed and the aerodynamic coefficients are compared with the wind tunnel results (Ahmed SR. et al., 1984) shown in table 1.

Figure 4. Hexa Structure Mesh

Table 1. Grid Independence of Simulation Results alongwith the Experimental Results

Wind tunnel Cd	0.285				
Tetra-prism mesh	**No of cells**	**C_D value**	**Hexa mesh**	**No of cells**	**C_D value**
coarse	542,158	0.374(+31.2%)	**Coarse**	391,276	0.331(+16.35%)
medium	1,055,875	0.299(+4.91%)	**medium**	1,018,752	0.308(+8.10%)
fine	3,559,042	0.293(+2.17%)	**Fine**	2,361,640	0.308(+8.10%)

Comparing the drag coefficients error values, it is obvious that the grid independence is achieved for both types of meshing. In both cases, the fine meshes show the lowest level of error than the medium level mesh. Therefore, the fine mesh is set as the baseline considerations for all further analysis.

4. DISCUSSION OF RESULTS

In this section, the numerical results computed for all the cases are presented and discussed. For the assessment and the performance of the different numerical schemes the profiles for Velocity and turbulent wakes and estimated drag and lift coefficients are obtained.

Drag and Lift Coefficients are defined as

$$C_D = \frac{2F_D}{A_s V_\infty^2} C_L = \frac{2F_L}{A_s V_\infty^2}$$

Where$C_D C_L$are the drag and lift coefficient respectively; ⍰is the projected area of the Ahmed model; $V_\infty^2$is the free stream velocity from the x-direction; andF_D, $F_L$are the drag and lift forces respectively.

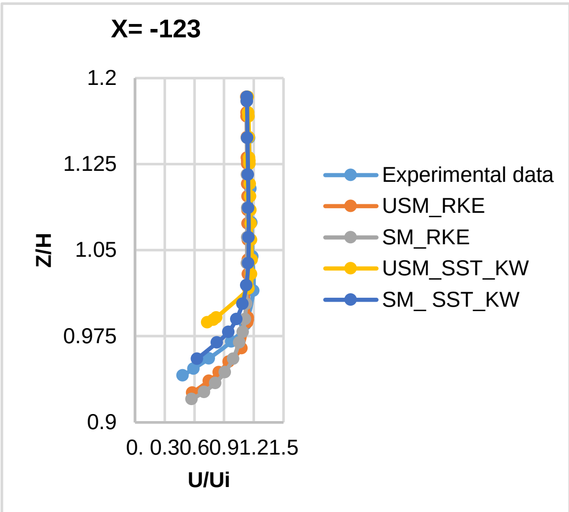
Table 2. Comparison of the present values of CD andCLwith the existing Experimental & Numerical Results

Source	Procedure used	Reynolds number	C_D value	C_L value
(Ahmed SR. et al., 1984)	Experimental	4.29 x10^6	0.285	-
Guilmineauet.al ,2018	K-Omega SST	0.768x10^6	0.322	0.173
Mishra et. al , 2017	Standard k-epsilon	0.768x10^6	0.275	-
Uddin. Met.al,2018	AKN k-epsilon	1.7x10^6	0.357	0.272
Present (tetra-prism mesh)	SST_KW	0.768x10^6	0.331	0.282
Present (Hexa-mesh)	SST_KW	0.768x10^6	0.330	0.238
Present (tetra-prism mesh)	RKE	0.768x10^6	0.291	0.255
Present (Hexa-mesh)	RKE	0.768x10^6	0.308	0.289

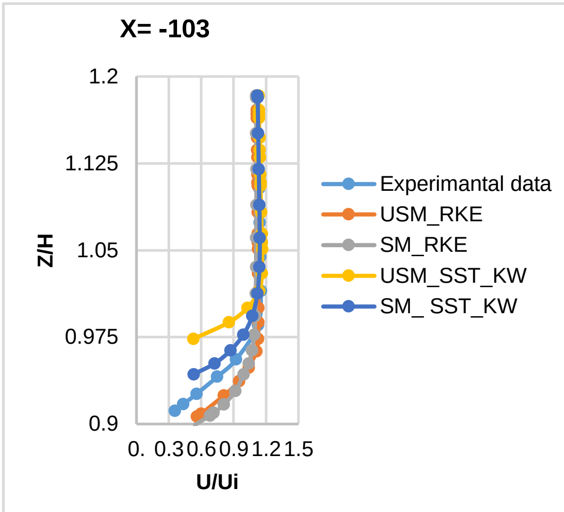
The results obtained from the RKE and SST k-ωturbulent schemes with different meshing structures such as Tetra-prism (TP)and Hexa-prism (HP)are summarized in table 2 and been compared with previous studies. It can be clearly shown that in all the cases,the drag and lift coefficients predict a similar with small variations from the existing results. The SST k-ωmodelling (Menter, 1994)provides an improvement in the results in both Hexa-meshing (HP_SST k-ω)and tetra-prism mesh (TP_SST k-ω), since it has the capability to captures the necessary details of the flow that are generated near the wall as well in the far fields. In the other hand, RKE,with both the meshing as (TP_RKE) and(HP_RKE), uses the wall functionapproximation to capture the near wall details.

4.1 Velocity Profiles

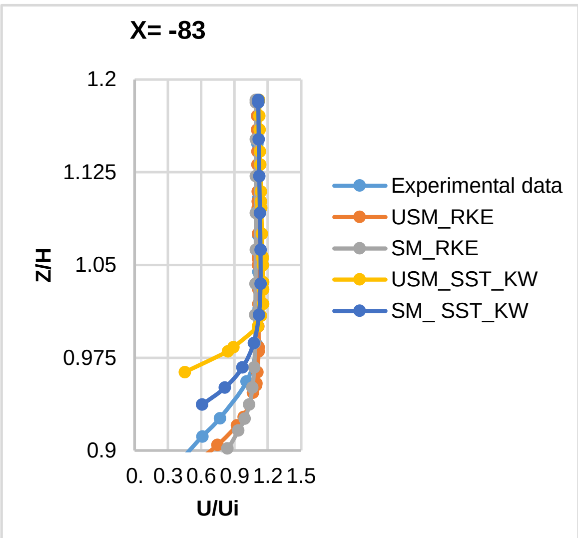
The flow behaviourdepends upon the slant angle applied over the geometry. Presently, the25^{o} angle slant Ahmed body lies in the critical angle range(Luca Brondolo, 2011) produces two counter-rotatingvortices in the side edges at the top of the slant surface. The flow separates over the slant and gets re-attaches again towards the end surface of the slant.



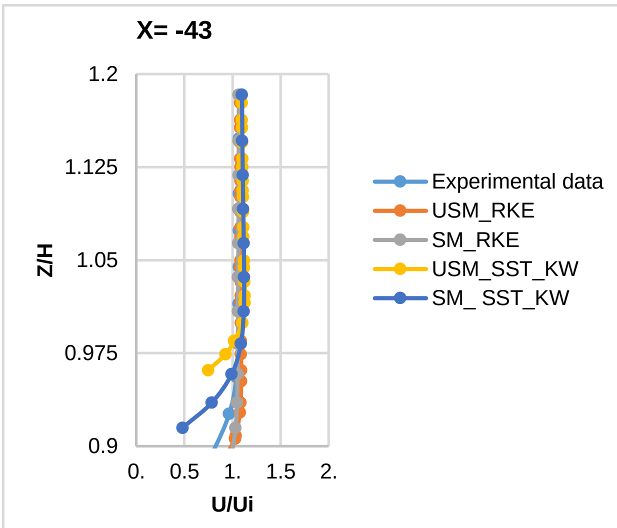
A



B



C



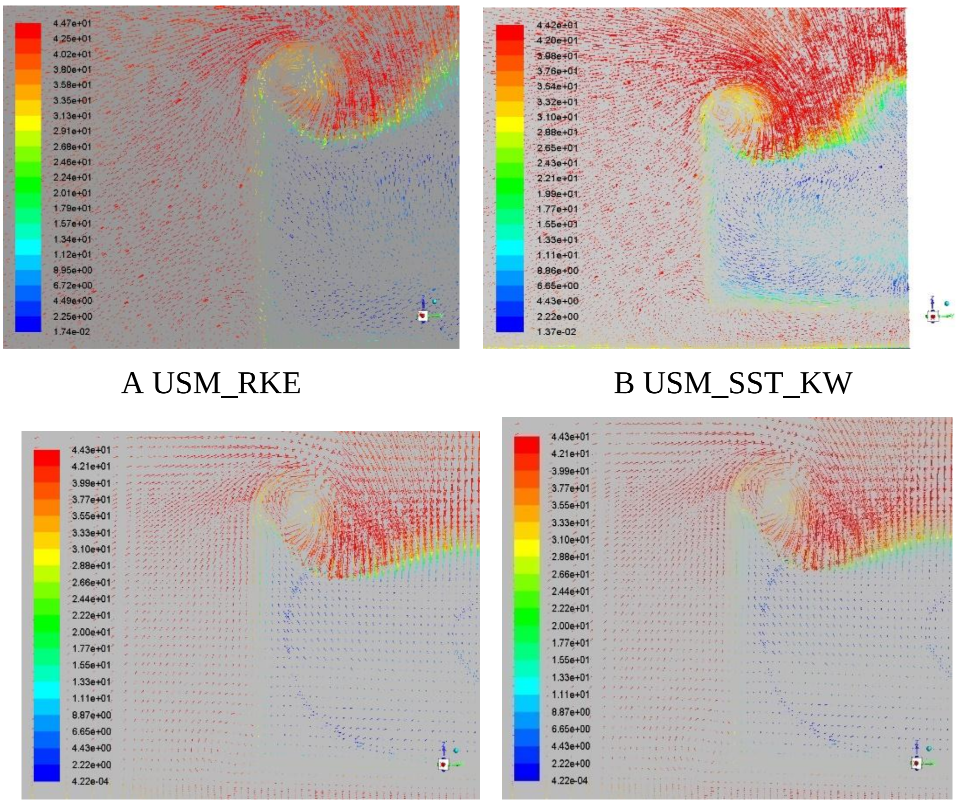
D

Figure 5. From A to D shown the velocity profiles lie over the slant at different positions

The profiles for the mean flow along height in the centre plane at different stream wise locations are taken according to the available experimental data (ERCOFTAC). It is concluded that; the results predicted by turbulence models are very poor compared with that of existing experimental results. But SM_ SST_KW modelling generates a similar trend as that of the experimental data. It also shown that from all the modelling that USM_RKE and SM_RKE gets over predicting results whereas for the USM_SST_KW and SM_ SST_KW generates under predict profile values.

4.2 Turbulent Wakes

The Iso-velocity surfaces for the four types of wake flow structures taken at x=80mm for the 25^{o} slant Ahmed body are shown in Figure 6. From the wake structure's, it can be observed that, the flow gets separation from the wall surface which tends to produce rotating vortices in the both side edges of the body. There is also a reverse flow in the mid-zone of the wake structures. This occurs due to the influence of the adverse pressure gradient generating the vorticity. All turbulent models show the vorticity effects at the side's wall of the slant surface. It is also been seen that the wake intensities are well captured in structure meshing than with that of unstructured meshing for both turbulent models.



A USM_RKE B USM_SST_KW

C SM_RKE D SM_ SST_KW

Figure 6. Iso-Velocity of the turbulent wakes structures for different turbulent schemes at x=80 mm.

5. CONCLUSION

The following results are drawn from this investigation employing Ansys fluent simulations with eddy viscosity turbulence schemes, RKE and SST k-ω, applied to the computational domains of hexa-mesh and tetra- prism grids over the Ahmed body.

i. Both RANS methods are capable of predicting the unstable time-average flow characteristics that have been applied to the hexa- and tetra-prism grid elements.

ii. The values of drag and lift coefficients obtained using hexa-prism and the SST k-scheme differ slightly from those obtained previously.

iii. The SST k-scheme is capable of capturing the intricacies of the flow fields close and far from the wall, whereas the RKE scheme uses near wall function approximations.

iv. In all approaches, the hexa-prism mesh produces more consistent, convergent, and stable flow outcomes while requiring less memory and processing time than the tetra-prism mesh.

REFERENCES

[1] Aljure, D. E., Lehmkuhl, O., Rodríguez, I., & Oliva, A. (2014). Flow and turbulent structures around simplified car models. Computers and Fluids, 96, 122–135. https://doi.org/10.1016/j.compfluid.2014.03.013

[2] Ashton, N., West, A., Lardeau, S., &Revell, A. (2016). Assessment of RANS and DES methods for realistic automotive models. Computers and Fluids, 128, 1–15. https://doi.org/10.1016/j.compfluid.2016.01.008

[3] Fares, E. (2006). Unsteady flow simulation of the Ahmed reference body using a lattice Boltzmann approach. Computers and Fluids, 35(8–9), 940–950. https://doi.org/10.1016/j.compfluid.2005.04.011

[4] Gaylard, A. P., Kabanovs, A., Jilesen, J., Kirwan, K., & Lockerby, D. A. (2017). Simulation of rear surface contamination for a simple bluff body. Journal of Wind Engineering and Industrial Aerodynamics, 165(February), 13–22. https://doi.org/10.1016/j.jweia.2017.02.019

[5] GuilmineauE.. (2008). Computational study of flow around a simplified car body.Journal of Wind Engineering and Industrial Aerodynamics,96, 1207–1217. https://doi.org/10.1016/j.jweia.2007.06.041

[6] Guilmineau, E., Queutey, P., &Visonneau, M. (2018). Assessment of Hybrid LES Formulations for Flow Simulation Around the Ahmed Body. Computers and Fluids, (January). https://doi.org/10.1016/j.compfluid.2017.01.005

[7] http://www.ercoftac. org/fileadmin/user_uplo ad/bigfiles/sig15/data base/index. html.

[8] International, S. A. E., & Transactions, S. A. E. (2018). Some Salient Features of the Time -Averaged Ground. 93(1984), 473–503.

[9] Le Good, G. M., & Garry, K. P. (2004). On the use of reference models in

automotive aerodynamics. SAE Technical Papers, 2004(724). https://doi.org/10.4271/2004-01-1308

[10] Lienhart, H., & Becker, S. (2003). Flow and turbulence structure in the wake of a simplified car model. SAE Technical Papers, 323–330. https://doi.org/10.4271/2003-01-0656

[11] Luca Brondolo,Comparative Investigation Of Large Eddy Simulation And Rans Approaches For External Automotive Flows.Master Thesis,(2011). (July), 2009–2011.

[12] Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32(8), 1598–1605. https://doi.org/10.2514/3.12149

[13] Mishra, S., Misra, A., Rao, P. S. V. R., Rao, D. N. (2019). MULTIPHASE MIXTURE MODEL SIMULATION OVER A SIMPLIFIED CAR. Advances and Applications in Fluid Mechanics,23(1), 17–24.

[14] Mishra, S.,Misra, A.,Rao, P. S. V. R.,Rao, D. N., (2017). Simulation ofAerodynamic Flow Parametersovera Simplified Sedan Car.International Journal of Engineering, Science and Mathematics, 6(8),1265-1274.

[15] Moghimi, P., &Rafee, R. (2018). Numerical and experimental investigations on aerodynamic behavior of the Ahmed body model with different diffuser angles. Journal of Applied Fluid Mechanics, 11(4), 1101–1113. https://doi.org/10.29252/jafm.11.04.27923

[16] Morgut, M., & Nobile, E. (2012). Influence of grid type and turbulence model on the numerical prediction of the flow around marine propellers working in uniform inflow. Ocean Engineering, 42, 26–34. https://doi.org/10.1016/j.oceaneng. 2012.01.012

[17] Serre, E., Minguez, M., Pasquetti, R., Guilmineau, E., Deng, G. B., Kornhaas, M., …Rodi, W. (2013). On simulating the turbulent flow around the Ahmed body: A French-German collaborative evaluation of LES and DES. Computers and Fluids, 78, 10–23. https://doi.org/10.1016/j.compfluid.2011.05.017

[18] Tientcheu-Nsiewe, M., Tcheukam, D., Toko, –, Murzyn, F., Mehel, A., &Patte-Rouland, B. (2016). Ahmed body, Wake, Velocity field, Turbulent flow, CFD; Ahmed body, Wake, Velocity field, Turbulent flow, CFD. American Journal of Environmental Engineering, 6(6), 157–163. https://doi.org/10.5923/j.ajee.20160606.01

[19] Uddin, M., Mallapragada, S., &Misar, A. (2018). Computational Investigations on the Aerodynamics of a Generic Car Model in Proximity to a Side-Wall. SAE Technical Papers, 2018-April, 1–10. https://doi.org/10.4271/2018-01-0704.