IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e- ISSN: 2278-1684, p-ISSN : 2320–334X
PP 01-05
www.iosrjournals.org

NUMERICAL ANALYSIS FOR DRAG REDUCTION IN COMMERCIAL BUSES

J. Godwin John[1], A.Muthuvel[2], N. Prakash[3]
[1](M.Tech ICE, HITS/ HINDUSTAN University, INDIA)
[2, 3](Automobile Department, HITS/ HINDUSTAN University , INDIA)

ABSTRACT: *Deterioration of fuel, demand for crude oil, worldwide extreme consumption etc. are the various reasons for the fuel demand and which leads in fuel price rising. An alternate way for the fuel consumption is assumed to be the reduction of drag. A small modification in the exterior design of the vehicle is done to carry out the study. Once the external body is modified an utmost care is taken in running the simulation by considering many factors such as mesh, boundary conditions, turbulence models etc. In this work numerical analysis of bus is carried out for the nine turbulence models to find the suitability of the turbulence models. Out of this AKN k-ε turbulence model predict the aerodynamic coefficient in the expected range when compared with the experimental data.*

Keywords – *Aerodynamics, Bus, Drag reduction, Fuel consumption, Turbulence models.*

I. INTRODUCTION

Public transports like bus and trucks are one of the major contributions by the automobile industry towards the mankind of the world. Latest research fields lies on the exterior structure of the bus for its modern look which basically explores the aerodynamics. Real workers and researchers believed that by modifying the outside design, projecting the frontal area, using drag reduction devices are the wise way to improve the fuel efficiency by reducing the drag force.

In a bus, the resistance due to the aerodynamics plays a vital role at higher speeds. When the design of the bus body is very flat the resistance is more and hence the researchers gave attention towards the design of the bus body. It is said that about 60-70 percent of the total wind-averaged drag of a bus is attributed to pressure loads acting on the vehicle fore body making it the principal area for drag reduction [1]. The power required to overcome the resistance is very high, hence it is sure that aerodynamics is an important factor to be considered. From many reviews it is clear using add on components would reduce the drag force. **ArunRaveendran** et al. [2] worked on his research changing the body design by lowering the body of the bus and reduced the drag coefficient up to 0.29 from 0.53. **Subrata Roy** et al [3] modified the design of the truck by this 30% of drag reduced and fuel is consumed around 35%. **Arthur E.Sheridan** et al. [4] did the study in the van body trailer adding the drag reduction devices. Here flow vanes are used so that the flow will become smooth and the drag is reduced. **Amit Ray et al.**[5] carried out CFD analysis of AUV bare – hull with Fluent software. For the analysis they used standard *k*-ε, Realizable *k*-ε, RNG *k*-ε, standard *k*- ω and SST *k*-ω model to find the hydrodynamics coefficients with various drift angles. Based on their study, the authors concluded that Realizable *k*-ε model predicts better than other models. The computed value of axial force is within 20% for a drift angle greater that 4 degree. **He Zhang et al.**[6] carried out numerical simulation of AUV vehicle by CFD software Fluent to evaluate manoeuvrability of vehicles and the corresponding hydrodynamic coefficients. Based on the study, the authors concluded that the computer simulation provides a low-cost reliable data for the hydrodynamic tests.**M. M. Karim et al.**[7] carried out the analysis of 2-D axisymmetric flow around bare submarine hull using both structured and unstructured grid. Shear Stress Transport (SST) k-ω model is used to simulate turbulent flow past the hull surface. They observed that unstructured grid gives better result and flow visualization than the structured grid. The ultimate aim of the project is to reduce the drag and computational time which has the huge impact on fuel consumption and research and development cost.

II. DIMENSIONAL SPECIFICATIONS

As the existing bus has a flat front it has more resistance to air and hence the drag force is very high and the stability is also very low at higher speed. In order to overcome these problems, we have come up with our exterior body design keeping the existing model as benchmark. Here we have made four models each having different exterior body styling and appearance. Here we have designed four different buses, in order to

optimize the contribution of each modification on the drag force. The first bus is the existing bus. The second bus has side tapering towards the rear end to provide better streamlined flow. The third bus has an aerodynamically shaped front with front spoiler and the remaining is same as that of first bus. The fourth bus has as aerodynamically shaped front with front spoiler. It also has a side and roof tapering towards the rear end. Commercial software SOLID WOKS (2010) has been utilized to design the bus model with its actual dimensions scaled to 1:30. With the wheel base as 6200 mm, front over angle as 2026 deg or mm, rear over angle as 3474 deg or mm, Overall length as 11700 mm, Front track as 2020 mm, Rear track as 1816 mm, Minimum ground clearance as 260 mm, Turning radius as 9025 mm, Gross vehicle weight is 16200 kg or mm.
The Wind Tunnel analysis and numerical analysis of flow over the bus is made for various speeds with all the buses from 1-4. Drag forces are measured and compared for various speeds from 80 to 115 kmph with an increment of 5kmph [8].

III. NUMERICAL ANALYSIS

Flow over the bus is numerically analyzed with Reynolds Averaged Navier-Stokes Equation in the commercial solver Star-CCM+. After importing the three dimensional model, Unstructured Polyhedral mesh is used and the various turbulence models like Standard k-ε model, Standard k-ε two layer model ,Standard k-ε low Re model, Realizable k-ε model, Realizable k-ε two layer model, AKN k-ε model, V2F k-ε model, SST Menter k-ω model and Standard k-ω Wilcox are used. Based on the overall results AKN k-ε model gave good result. Henceforth it is used for further analysis. While running the simulation in order to avoid divergence the first 300 iteration is performed under first order upwind scheme with relaxation factor of velocity with 0.3 and pressure with 0.1, the rest will simulate with relaxation factor of velocity with 0.5 and pressure with 0.3 until the force become steady. The convergence criteria used is 10^{-6} for every simulation. These facts are considered for providing good accuracy and convergence.

IV. RESULTS AND DISCUSSION

4.1 Grid independent test

The mesh sizes were increased from 1.15 to 1.75 (i.e., 1.15, 1.31, 1.57, 1.75) and the grid independent test is carried out for the bus 1 and bus 4. While running the mesh independency AKN k-ε two layer model is used. For the bus 1 and bus 4, 1.75million cell is considered as grid independent mesh as the deviation between 1.57 million and 1.75 million cells is less than 5%. The Difference between the Experimental and the analytical results were very less in the 1.75 million grids and henceforth it is selected for further simulation of bus 1 is shown in TABLE 1.

Table 1 Drag Force for various grids at 80kmph with AKN k-ε turbulence model.

Bus Number	No of grids in Millions	Pressure force N	Shear Force N	Drag Force N	Experimental Drag force N	% of Deviation
1	1.15	866	79	945	882.0	07.14
	1.31	832	83	916		03.85
	1.57	828	77	907		02.83
	1.75	804	83	887		00.65
4	1.12	517	102	620	705.6	12.13
	1.51	491	100	592		16.09
	1.57	513	101	614		12.98
	1.75	519	101	620		12.13

V. VARIOUS TRUBULENCE MODELS

The axial force values obtained for the various turbulence models, with unstructured mesh for drift angle 0^0 i.e. flow coinciding with the bus1 and bus 3 axis are tabulated in Table 2 and 3. The results of the present analysis is compared against the experimental data and the results obtained by Muthuvel et al. [8]. It can be seen that V2F k-ε and AKN k-ε model predicts better than other models. For AKN k-ε model, the deviation is 0.65%, for V2F k-ε model 3.32% whereas for standard k-ω model it is 3.5% for bus 1 and For AKN k-ε model, the deviation is 12 %, for V2F k-ε model 13 % whereas for standard k-ω model it is 9 % for bus 4.

Table 2 Drag force for BUS 1 at 80 kmph

Number of grids in Millions	Pressure force N	Shear force N	Total drag force N	Experimental drag force N	Percentage of deviation %	Various turbulence models
1.75	0804.59	083.22	0887	882	00.65	AKN k-ε model
	1077.44	108.46	1185	882	34.45	Realizable k-ε model
	1077.32	108.43	1185	882	34.43	Realizable k-ε model two layer
	1596.83	110.94	1707	882	93.62	Standard k-ε model two layer
	1592.20	110.91	1703	882	93.09	Standard k-ε model low Re
	1590.37	110.91	1701	882	92.89	Standard k-ε model two layer
	0823.97	087.29	0911	882	03.31	V_2F k-ε model
	0851.77	87.73	939	882	06.51	SST (Menter) k-ω model
	0825.59	88.03	913	882	03.58	Standard Wilcox k-ω model

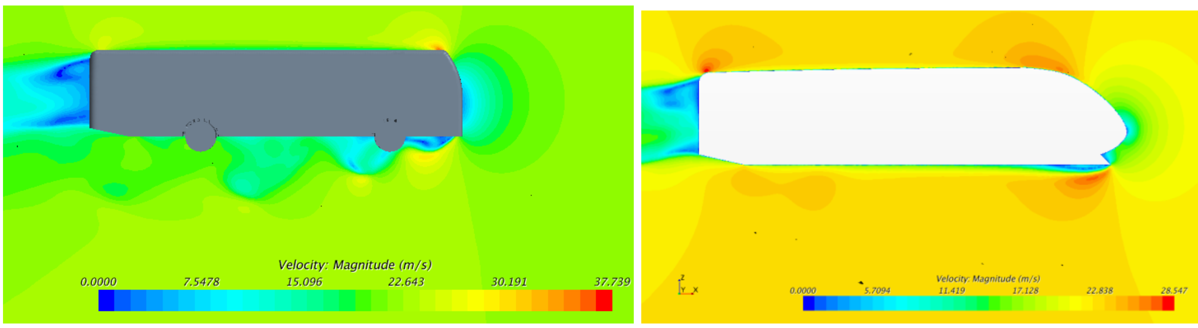
Table 3 Drag force for BUS 4 at 80 kmph

Number of grids in Millions	Pressure force N	Shear force N	Total drag force N	Experimental drag force N	Percentage of deviation %	Various turbulence models
1.75	519.12	101.46	640.5	705.6	12.13	AKN k-ε model
	589.09	127.64	716.7	705.6	01.57	Realizable k-ε model
	589.28	127.59	716.8	705.6	01.59	Realizable k-ε model two layer
	747.29	137.68	884.9	705.6	25.42	Standard k-ε model
	747.81	137.64	885.4	705.6	25.49	Standard k-ε model low Re
	747.79	137.64	885.4	705.6	25.48	Standard k-ε model two layer
	503.41	106.57	609.9	705.6	13.55	V_2F k-ε model
	518.84	105.49	624.3	705.6	11.51	SST (Menter) k-ω model
	529.66	107.37	637.0	705.6	09.71	Standard Wilcox k-ω model

Contours, Path lines and Vectors diagram:

The velocity contour diagram for AKN k-ε Low Re layer model is shown in Fig1.

Bus 1 **Bus 4**



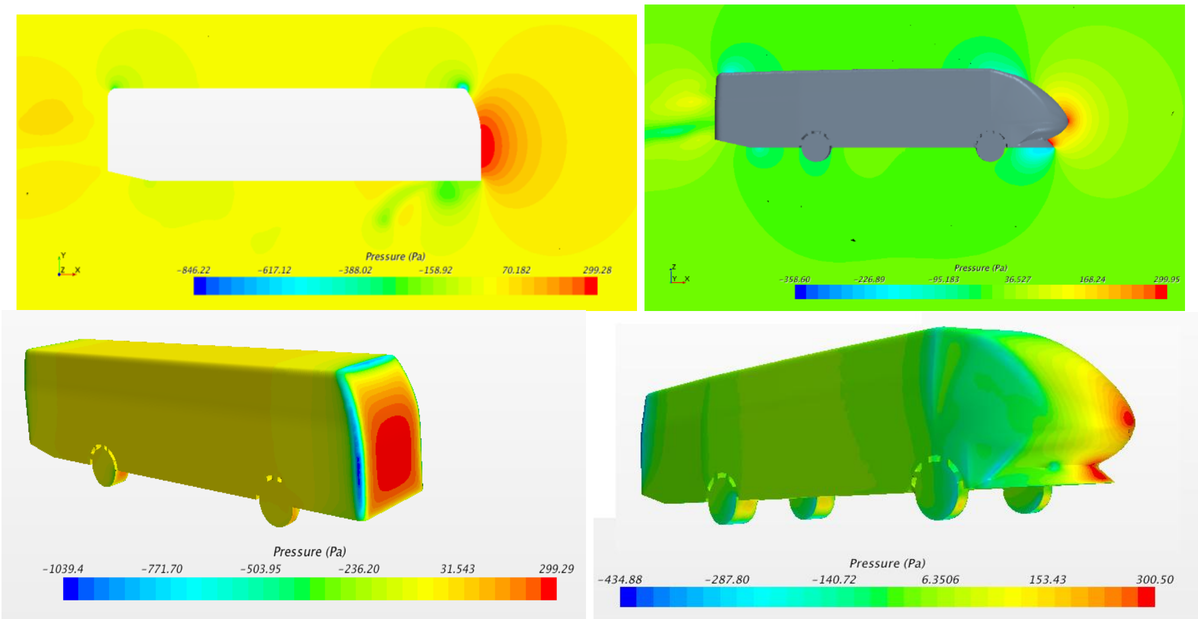


Figure 1 Velocity and pressure contour for AKN k-□ layer model

From the velocity contour diagram, the variation of the velocity of flow over the bus 1 and 4 is seen. At the front part of the bus, stagnation condition is clearly captured. Also, boundary layer formation is seen near the walls. As the flow advances over the bus 1, the velocity increases gradually and then reduces in the rear region due to curvilinear nature of bus with bus 4.Path lines of flow over the Bus on 1 and 4 are shown in Figs 3 and 4.

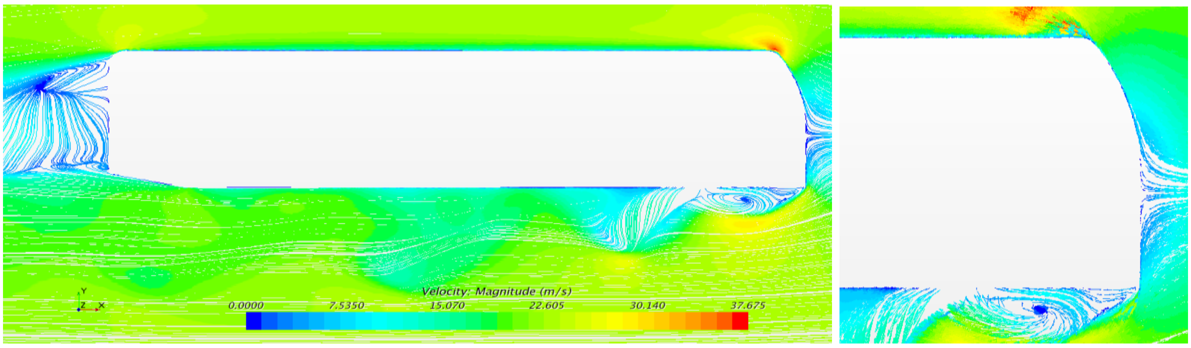


Figure 2 Path lines on Bus 1

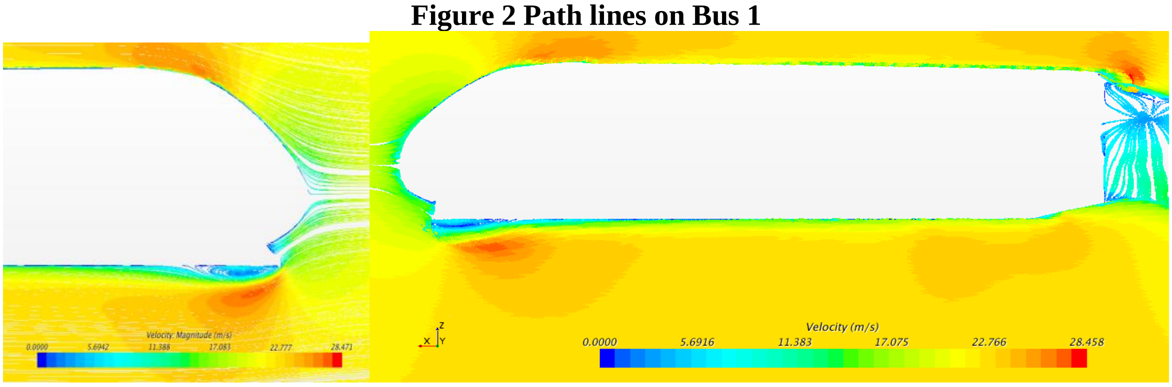


Figure 3 Path lines on Bus 4

VI. CONCLUSION

In this paper, analysis is carried out for AL PSV 4/157 model of bus 1 and modified bus 4 with commercial solver STAR-CCM+. Analysis is carried with unstructured polyhedral meshes for 0 degree drift angles. In the analysis, various turbulence models, viz. realizable k-ε two layer models, Standard k-ε low Re model (buffer layer), Standard k-ω (Wilcox) model, standard k-ε model, k-ω SST model (buffer layer), V2F k-ε model (buffer layer)and AKN k-ε model (buffer layer) used. It is seen that out of all the turbulence models used, AKN k-ε model gives better results than any other models.

REFERENCES

[1] Jeff Patten et al, (2012), "Review of Aerodynamic Drag Reduction Devices for Heavy Trucks and Buses", CSTT-HVC-TR-205, National Research Council of Canada, Centre for surface transportation technology.
[2] ArunRaveendran et al, (2009), "Exterior Styling of an Intercity Transport Bus forImproved Aerodynamic Performance", SAS TECH, Volume 8 issue 2.
[3] Subrata Roy and Pradeep Srinivasan, (2000), "External flow analysis of a truck for Drag Reduction", SAE 2000-01-3500.
[4] Arthur E. Sheridan and Steven J. Grier, (1978), Drag Reduction Obtained by Modifying a Standard Truck".
[5] He Zhang, Yu-ruXu and HaopengCai, "Using CFD software to calculate Hydrodynamic coeffecients", Journal of Marine Science App, Vol. l9, pp 149-155, 2010.
[6] M. Karim, M. Rahman and M. A. Alim, "Computation of turbulent viscous flow around submarine hull using unstructured grid", Journal of Ship Technology, Vol.5, No.1, pp. 38-52, (January 2009), ISSN: 0973-1423.
[7] Amit Ray, S N Singh and V Seshadri, "Evaluation of Linear and Non-linear Hydrodynamic coefficients of Underwater vehicles using CFD", OMAE2009-79734, Proceedings of ASME 28th International Conference on Ocean, Offshore and Arctic Engineering, OMAE2009, Honolulu, 2009.
[8] A.Muthuvel, M.K.Murthi, Sachin.N.P, Vinay.M.Koshy, S.Sakthi, E.Selvakumar "Aerodynamic Exterior Body Design of Bus" in International Journal of Scientific & Engineering Research, Volume 4, Issue 7, June 2013 ISSN 2229-5518.