Drag Reduction in Cars Using Add on Devices

¹Velmurugan G, ²Nihar Farzaana F, ³Sopiya K, ⁴Priyadharshini C, ⁵Roselin Madhumitha P, ⁶Safrin A

¹Assistant Professor, ^{2,3,4,5,6}Final year student Department of Aeronautical Engineering, Excel Engineering College, Affiliated to Anna University, Tamil Nadu, India.

Abstract: Of the globally available fuel resources 77% is taken by ground automobiles, which accounts for nearly 40% of global air pollution and emission of greenhouse gases. Ultimately drag being the main reason for fuel consumption. Based on study 15% reduction in drag at a highway speed of 55mph can result in 5-7% of fuel savings. The prime methodology of the project is to attach a device to the existing vehicle and analyse the wake region at the rear end, using the numerical model k-Epsilon in the Ansys Fluent 17 platform. Two-dimensional model of the majorly used cars are generated without any modifications using Catia V5 modelling software. Then the same model is designed with an add on device. Flow around both unmodified and modified vehicles are analysed and compared for drag coefficient. A rectangular enclosure is constructed in the Design Modeller and the model is subtracted with Boolean operation. The domain is finely meshed. Boundary conditions are given in accordance to the road transport environment, calculations are done with velocity inlet and pressure outlet. The graphs are plotted for Cd and Cl. The contours are then recorded at each trial for pressure and velocity. The results are then compared. There occurs a significant reduction of about 1.29 to 0.98 in drag coefficient which is about 24% of decrease of drag from the existing vehicle to the modified vehicle.

Index Terms: Drag induced fuel consumption, Wake behind bluff body, flow analysis, Ansys fluent.

I. INTRODUCTION

To achieve the above-mentioned objectives, the work is split into various stages. Initially, modeling of the flow around a car within range of Reynolds number is done by using basic steady state simulation methods. This is done as a pilot study for the further application of CFD on the computation of a more complex flow using advanced CFD techniques at a later stage of the work. The next stage is the simulation using the unsteady and more advanced LES model on the flow around a car to study the vortex shedding phenomenon in the wake region of the flow. This acts as a first step towards the investigation of the effect of vortices on bluff body flow. The simulation of flow around rear sections investigates the changes of the drag coefficient with the increase in velocity. Apart from that validation of the turbulence models (LES and DES) at higher Reynolds number of for the flow around a car body is conducted. This ensures that the turbulence models are capable of capturing the flow characteristics accurately not only at high Reynolds number but also at high speed. The work on the simulation of flow provides a general idea of the flow patterns and the expected outcomes on the flow, numerical modelling techniques of the simulation are then applied on the flow around vehicle body to investigate the wind effect on it. The work proceeds like

A. Modelling and flow analysis:

- Modelling and flow analysis of unmodified rear body of car Model-A
- Modelling and flow analysis of modified rear body of car Model-B
- Modelling and flow analysis of modified rear body of car Model-C
- Modelling and flow analysis of unmodified sedan car Model-D
- Modelling and flow analysis of modified sedan car **Model-E**

B. Comparison of Cd plots, Cl plots and contours.

- Comparison of speed vs. Cd plot of Model-A, Model-B, Model-C
- Comparison of speed vs. Cl plot of Model-A, Model-B, Model-C
- Comparison of speed vs. Cd plot of Model-D and Model-E
- Comparison of speed vs. Cl plot of Model-D and Model-E
- Comparison of velocity and pressure contours obtained.

C. Conclusion

The results are compared to find the model with least coefficient of drag and the pressure and velocity contours are compared to find the model which generates the smaller area of wake region.

II. DESCRIPTION

The profile of the majorly used cars have been taken and modelled in Catia V5. For example, like Maruthi alto 600 and Maruthi 800. The models have been exported to Ansys Fluent and constructed an enclosure around it to study the drag coefficient around them. The details of operation done on Design modeler are as given in Table 1, pictorial representation of the dimensions and 3D model are shown in Figure 1, Figure 2.

S.no	Geometry details	Input value	S.no	Mesh details	Input value
1.	length of car body	3430 mm	8.	mesh relevance	40
2.	width of car body	1490 mm	9.	mesh method	all triangles
3.	height of car body	1475 mm	10.	mesh inflation	3 layers
4.	length of enclosure	6430 mm	11.	named selections	inlet, outlet, car body
5.	height of enclosure	4000 mm	12.	number of nodes	2860
6.	conversion to 2d	Mid Surf operation	13.	number of elements	5100
7.	creation of domain	Boolean subtract			

Table 1 Details of geometry and mesh of the domain



Figure 2 3d model of Alto 600



The required domain is constructed around the bluff body as shown in Figure 3 and Figure 4. The boundary conditions are fed to the Ansys Fluent module according to the general prevailing conditions as given in Table 2. The iterations are done until the solution converges, as in Figure 5 and Figure 6.

Table 2 Solu	tion and setup parameters	
S.no	Solution details	Input value
14.	solution launcher	double precision
15.	solution type	serial solver
16.	general setup	pressure based – steady state analysis
17.	model setup	energy on
18.	nature of flow	in viscid
19.	Reynolds number	104 and above
20.	density of fluid	1.225 kg/m^3
21.	boundary conditions	velocity inlet, pressure outlet
22.	velocity inlet	11.11 m/s
		13.88 m/s
		16.66 m/s
		19.44 m/s
		22.22 m/s
		25 m/s
		27.77 m/s
		30 m/s
		33.33 m/s
23.	pressure outlet	0 Pascal

hla 2 Calutia 1

24.	courant number	2.5 & 7.5
25.	y plus values	1 - 40
24.	solution method	pressure-velocity coupling
25.	solution scheme	simple
26.	gradient discretisation	least square cell based
27.	pressure discretisation	second order
28.	momentum discretisation	second order upwind
29.	energy discretisation	second order upwind
30.	solution monitors	coefficient of drag coefficient of lift
31.	solution initialisation	standard initialisation
32.	number of iterations	1000 per case
33.	reporting interval	1
34.	profile update interval	1
35.	contours obtained	pressure contour
		velocity contour



The contours of pressure at speeds 22.2m/s, 27.7m/s and 33.33m/s are shown in Figure 7-a, b and c respectively.









From the Graph1 and Graph 2, the co-efficient of drag for the unmodified vehicle-model A is found to be 1.40. The study is proceeded to lower this drag, by reducing the size of wake behind the body. So, the rear side of the body is modified by attaching a semi-spherical add on of radius 0.4 meters. The same is designed on Catia V5, imported to the workbench where, the domain is constructed as shown in Figure 8 and Figure 9. Meshing and boundary conditions are applied similar to the unmodified vehicle. They are represented in Figure 10,11



	An one spectra the second seco	H HH		500 500 500 500 500 500 500 500 500	Constant Section 2014 Constant Sect	Vertical Annual	
Security	08 =+ == >=		· ¢ (1)] 1] 1] 1] 1] 1] 1] 1] 1] 1]	1000	Control and a second seco	tere	

The velocity contour and pressure contour for Model B at a speed 33.33m/s is shown in Figure 12 and Figure 13 respectively.



Figure 13 Pressure contour at 33.33m/s



5.98e-01



From the Graph 3 and Graph 4, the co-efficient of drag for the modified vehicle-model B (radius 0.4m) is found to be 1.236. Which is considerably lower than the model A, for the same geometry, meshing and boundary conditions. The study is further proceeded for larger add on of radius 0.7 metres as shown in Figure 14 and Figure 15, with same conditions and the flow around it is analysed for drag that is for model C.



These Figures 16 and 17 represent the velocity and pressure contour of Model C at a speed of 33.33 m/s. The contours are later compared to obtain better insights of decrement in drag.

-3.68e+03

210



From the above Graph 6 and Graph 7, the drag co-efficient for the model C is found to be 0.98 which is lower than model A and model B. Now the study of add on is compared to the study of design modification. Add on could be fitted after manufacture, whereas design modification involves redesign and re modification. Let the models be Model D and Model E.

III. DESIGN MODIFICATION- MODIFICATION IN A SEDAN MODEL

To know the effectiveness of add on drag reduction, we analyses flow around the sedan model and modify the sedan model via design aspect part of it. Later the percentages of the drag reduction can be compared to find whether the drag reduction is effective by add on devices or design modification. Design modification involves the modification of complete vehicle, that would result in high expenditure. Whereas, the add on modification would be affordable to majority of the customers. A rough sedan car is modeled in Catia V5 and imported to Ansys fluent to construct the domain of study. The boundary conditions are applied similar to the previous models A, B, C. The drag coefficients are determined and graph between Cd and CL are plotted against velocity to compare the results with those of Model A, B, C. Table 3 states the steps of modelling and analysis of the sedan model.



520







From the table:4 inference is made that, the add on procedure from Model A to Model C, has an evident decrease in drag of about 30%. Whereas the design modification procedure, from Model D to Model E has produced a decrease in drag of about 20.8%. Both the values may subject to change from model to model. But on comparison, the design modification procedure is highly complex, as to reduce the drag of the existing vehicle, one has to remanufacture the vehicle with a proper desired design. Here, the objective is drag reduction on existing vehicle, without any remanufacturing of the product. An add on device could be purchased by any customer in accordance to his or her car's design, merely a lay man work. But the redesign and remanufacture is highly costly and it is dealt with research and development agencies with huge figured funds. At the present situation, everyone can add such devices to their vehicle to lower the usage of fuel to a certain extent and prevent the emission of poisonous gases to the atmosphere we live in. Future work would be focused on the shapes and sizes of such add on devices and to study their efficiency in drag reduction.

REFERENCES

- Ram Bansal and R. B. Sharma Department of Automobile Engineering, RJIT BSF Academy, Tekanpur, India Mechanical Engineering Department, RJIT BSF Academy, Tekanpur, India. Journal of Aerodynamics, Volume 2014, Article ID 678518.
- [2] Vipul Kshirsagar, Jayashri V. Chopade, "Aerodynamics of High-Performance Vehicles," International Research Journal of Engineering and Technology (IRJET), Volume: 05 Issue: 03, Mar-2018.
- [3] F. R. Bailey and H. D. Simon, "Future directions in computing and CFD," AIAA Paper 92-2734, 1992.
- [4] H. Taeyoung, V. Sumantran, C. Harris, T. Kuzmanov, M. Huebler, and T. Zak, "Flow-field simulations of three simplified vehicle shapes and comparisons with experimental measurements," SAE Transactions, vol. 106, pp. 820–835, 1996.
- [5] Bahram Khalighi, S. Zhang, C. Koromilas General Motors R&D Centre, "Experimental and Computational Study of Unsteady Wake Flow behind a Bluff Body with a Drag Reduction Device."
- [6] Mohd Nizam Sudin, Mohd Azman Abdullah, Shamsul Anuar Shamsuddin, Faiz Redza Ramli, Musthafah Mohd Tahir, "Review of Research on Vehicles Aerodynamic Drag Reduction Methods," International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS Vol:14 No:02.
- [7] K. Ahmad, M. Khare, and K. K. Chaudhry, "Model vehicle movement system in wind tunnels for exhaust dispersion studies under various urban street configurations," Journal of Wind Engineering and Industrial Aerodynamics, vol. 90, no. 9, pp.1051–1064, 2002.
- [8] Z. Yang and B. Khalighi, "CFD simulation for flow over pickup trucks," SAE Paper 2005-01-0547, 2005.