🌐 ***United International Journal of Engineering and Sciences*** 🌐
(UIJES – A Peer-Reviewed Journal); ISSN:2582-5887 | Impact Factor:8.075(SJIF)
📓 *Volume 5 | Special Issue 1 | 2025 Edition*
National Level Conference on "Advanced Trends in Engineering Science & Technology" – Organized by RKCE

Computational Fluid Dynamic Analysis of Aerodynamic Drag Reduction and Improve Fuel Economy

B JASWANTH[1], P CHARAN RAVI KISHOR[2], S UMA KRISHNA[3], V HARSHA VARDHAN[4]
[1234]UG Students, Dept of Mechanical Engineering, R K College of Engineering, Vijayawada, India
jaswanthroopus@gmail.com[1], charanravikishor@gmail.com[2], somarajuumakrishna49@gmail.com[3], Harsha56527@gmail.com[4]

Abstract – **In recent years, fuel quality in heavy trucks has become a growing concern for production and construction engineers. Enhancing vehicle aerodynamic efficiency is one of the most effective methods to reduce fuel consumption. This can be achieved by lowering the drag coefficient, which is directly linked to fuel usage. Engineers are exploring various factors, such as engine settings, weight, rolling resistance, and aerodynamic drag, to improve heavy vehicle performance. In this project, the frontal design of the container was modified to enhance aerodynamic performance. Computational modelling was conducted at different speeds (15 km/h, 45 km/h, 60 km/h, 80 km/h, and 100 km/h) to assess the impact of changes to the container's frontal area in heavy vehicles. Specific truck shapes were designed using CATIA V5, and ANSYS CFX was employed to simulate the results. Comparative studies were carried out, and the results showed that the proposed model demonstrated significant improvements in reducing drag and fuel consumption compared to the baseline model.**

I. Introduction

CFD is a computer-based method for modelling the behavior of systems including fluid flow, heat transfer, and other physical processes. It operates by solving (in a specific form) the equations of fluid flow across a region of interest with defined (known) boundary conditions.

The main objective of drag reduction is to obtain better fuel consumption of the truck and increase in the operational performances. This project is about the study of modelling and flow analysis of heavy truck with two different modified truck designs. It is also concerned with validation of simulation results with wind tunnel experimental results, concern with calculations of fuel consumption, power required to overcome drag, energy consumption by the truck and aerodynamic efficiency. The truck designs were developed in CATIAV5 R20 with in standard geometry. And the analysis was carried out for the designed model at speed 100kmph by using CFD.

In this study numerical simulations of different trucks configurations are performed. Starting with evaluating the results of the baseline simulation, configurations of interest can be identified. For each configuration the CFD process described below is performed. The CFD-process can be divided into three steps; pre-processing, solving and post-processing.

II. LITERATURE REVIEW

[1] R.H. HEALD. (1933) investigated four models of car for its drag coefficient and compared it with the one which was 10 years earlier. He found out that elimination of fenders and other projections together with pronounced fairing of body of one model reduced the drag coefficient quite significantly.

An additional decrease in value of drag coefficient was observed by eliminating the windshield and fairing the whole body of car so as to resemble a thick air plane wing section.

[2] SHOBIT SENGAR ET AL. (2014) determined forces acting on three different segments of a vehicle, the Hindustan Ambassador, Lamborghini Aventador LP 700-4 and an F1 car, by testing their models in a Wind tunnel. A comparison is done between the three models for the best aerodynamic features. The scaled models are tested under different wind conditions in. It was found that the F1 car is the most aerodynamic amongst the three followed by the Aventador and then ambassador. The former two has this result due to its low-slung body which results in lower ground clearance. Also, the linings of the coupe help in channelizing the air when the vehicle is in motion which leads the air to the rear end where spoilers are provided which provides additional stability at high speeds.

[3] ABDELLAH AIT MOUSSA ET AL. (2015) worked on reduction of Aerodynamic Drag in generic trucks using geometrically optimized rear cabin bumps. They used a 1/10th scaled half model of a generic truck and added three equally spaced bumps on the top of the cabin surface. Thereafter they used Taguchi or Orthogonal array optimization method to study the effect of these bumps on drag.

[4] YINGCHAO ZHANG ET AL. (2009) the details of the virtual wind tunnel test simulation were narrated in this paper. Applying the virtual wind tunnel test aerodynamic drag coefficient, velocity contour and pressure distribution were got. Some advices to reduce aerodynamic drag of the design car were put forward. It was found that it is a simple, effective, convenient and fast way to do aerodynamic numerical simulation based CFD in the process of car styling.

[5] ABDUL KAREEM SH. MAHDI – OBEIDI ET AL. (2014) tested an open wheel race car made by students of Taylor's University in a wind tunnel and in ANSYS fluent. They studied the effect of Radiator air channel in drag optimization and compared both experimental and numerical results. They found out that increasing the angle of tilt of radiator channel from 36˚ to 72.5˚ results in reduction in drag to 0.563 from 0.619. Both results agreed without much deviation in results. There was only 7.7% between both results.

[6] J ABHINESH ET AL. (2014) conducted a CFD analysis of two Volvo intercity trucks. Model one being the existing truck model and second being the modified one. This they did in order to reduce the aerodynamic drag and fuel consumption. After the CFD analysis they found drag reduction of about 10%. The Original model's coefficient of drag was found to be 0.8 and for the modified model was found to be 0.7.

[7] FRANCESCO MARIANI ET AL. (2012) numerically tested a race car model which was made by the students of University of Perugia. Their main experiment was focused around changing the design of car nose so as to optimize the aerodynamic drag. They called the original model as "A" and modified as "B". In model B they added a front wing, modified the headrest, adopted an air extractor and added a wing on front tire.

[8] ASHFAQUE ET AL. (2014) discussed the Drag force analysis of car by using low speed wind tunnel. In the testing they used the Piton tube, manometer and solid object (airfoil). They calculated the velocity of fluid and drag & lift forces. They observed that design of low-speed open circuit wind tunnel is somewhat different to other wind tunnel. Its diffuser is flexible. The construction of machine is low cost and Design is very easy. It's using materials easily available in the market. The machine is useful to educational and research purpose.

[9] KEISUKE NISUGI ET AL. (2004) worked on reduction of Aerodynamic Drag for vehicle having feedback flow control. In their study they mounted a sensor (control flow nozzle) which provided the controller the information about velocity and pressure components. As per the requirement the controller drives the actuator which in turn operates the control port where blowing and suction of air takes place. The nozzle was placed in a portion of the front wind shield. Proper systematic calculation resulted in 20% drag reduction as compared to the vehicle without the feedback flow control system.

[10] L. ANANTHAN RAMAN ET AL. (2016) conducted a comparative study of different methods of aerodynamic drag reduction to reduce fuel consumption in vehicles. They conducted passive tests on a SUV model by extending its rear end (rear fairing), adding a rear plate (rear screen) and by adding a vortex generator (Delta wing and bump shaped). A 6.5% and 26% reduction of drag was found by installation of rear screens and rear fairing respectively. Among the vortex generators the delta wing type were found more effective drag reducers.

[11] UPENDRA S. ROHATGI ET AL. (2012) tested a small-scale model of General Motor SUV and tested in the wind tunnel for expected wind conditions and road clearance. Two passive devices, rear screen which is plate behind the car and rear fairing where the end of the car is aerodynamically extended, were incorporated in the model and tested in the wind tunnel for different wind conditions. The conclusion was that rear screen could reduce drag up to 6.5% and rear fairing can reduce the drag by 26%. It was also mentioned that efficiency of rear screens from point of view of drag reducing equally depends on configuration, dimensions and arrangement of screens as well as on model's rear part configuration.

[12] ROSE MCCALLAN ET AL. (1999) conducted Wind tunnel analysis of model of 1:14 Class 7 & and Class 8 heavy duty Sandia trucks to reduce their aerodynamic drag and so as to improve the fuel economy. The PIV (Particle image velocimetry) measurements were taken in the model wake. Oil film interferometer techniques (OFI) for measuring skin friction and pressure sensitive paint (PSP) measurements were also used. They found that PIV approach to calculate various parameters in the Wind tunnel can be effective in finding more precise and accurate results.

III. METHODOLOGY

In this study numerical simulations of different vehicle configurations are performed. Starting with evaluating the results of the baseline simulation, configurations of interest can be identified. For each configuration the CFD process described below is performed. The CFD-process can be divided into three steps; pre-processing, solving and post-processing.

This project is conducted by evaluating different drag reducing trailer devices by means of CFD. ANSYS and CATIA are the software used and the setup and essential settings are described. Further on, from the evaluation of the individual devices, combinations of these are tested and analyzed to see if drag can be improved even more.

IV. RESULTS AND DISCUSSIONS

3.1 Pressure Variations on Truck

Following are the results obtained by the CFD Fluent simulation analysis conducted on the heavy vehicle truck at an optimal speed of 15 KMPH. The results of pressure acting on the truck were plotted below for both basic as well as advanced (proposed) model.

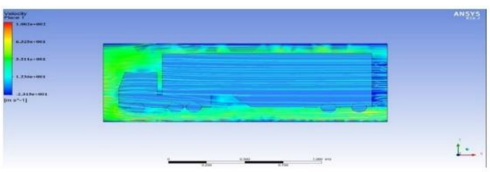


Figure: Pressure acting on the truck basic model @ 15 kmph

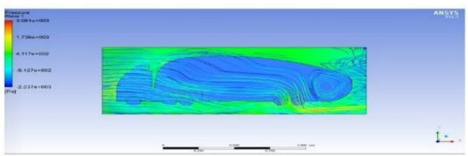


Figure: Pressure acting on the truck proposed model @ 15 kmph

3.2 Velocity Variations on Truck

These are the results obtained by the CFD Fluent simulation analysis conducted on the heavy vehicle truck at an optimal speed of 15 KMPH. The results of velocity acting on the truck were plotted below for both basic as well as advanced (proposed) model.

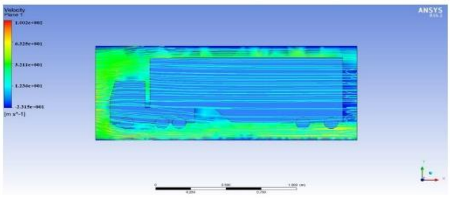


Figure: Velocity acting on the truck basic model @ 15 kmph

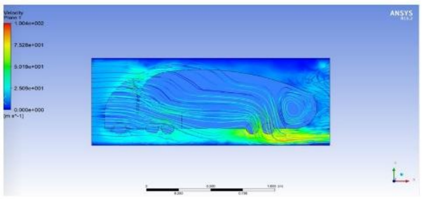
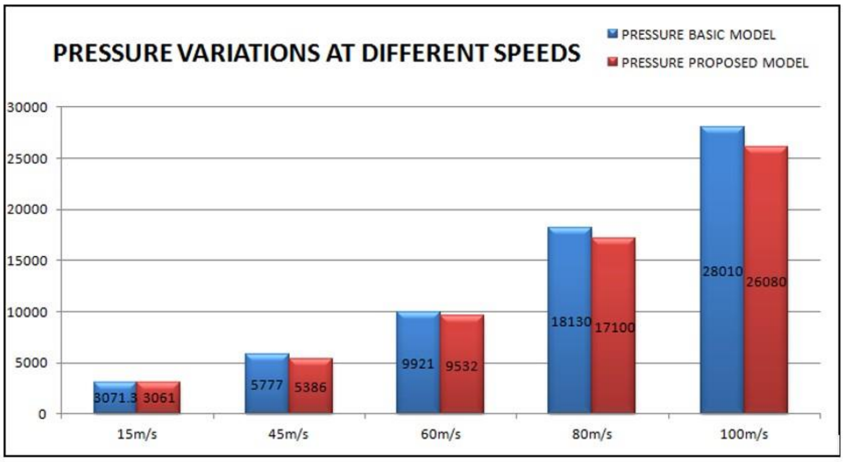


Figure: Velocity acting on the truck proposed model @ 15 kmph

3.3 Pressure Variations at Different Speeds

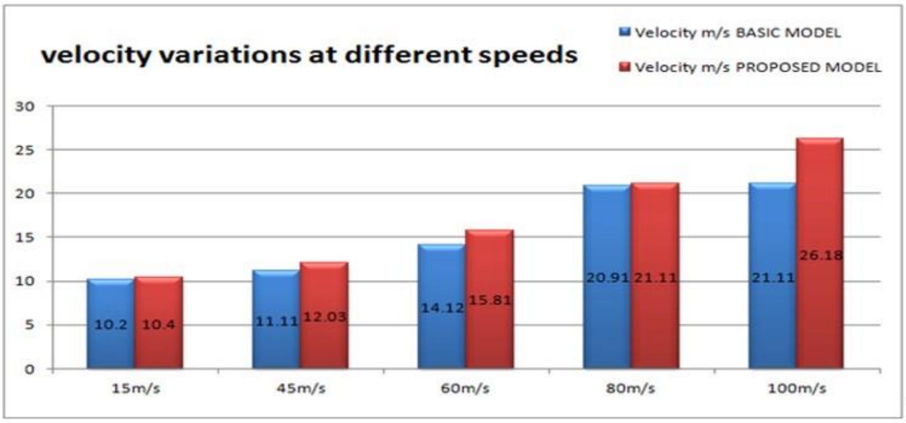
The following bar graphs illustrate the pressure variations at different speeds of truck with speed variations of 15kmph/45kmph/60kmph/80kmph/100kmph respectively. The maximum outcomes are shows at a value of a 100kmph. The pressure at 100kmph was 28010 Pa for existing basic model whereas 26080 Pa for the proposed model. It clearly shows the amount of pressure was reduced for the proposed model.



Graph 1: Pressure variation at different speeds for basic and proposed model

3.4 Velocity Variations at Different Speeds

The below graphs discussed the velocity variations at different speeds of truck with speed variations of 15kmph/45kmph/60kmph/80kmph/100kmph respectively. The maximum outcomes are shows at a value of a 100kmph. The velocity at 100kmph was 21.1m/s for existing basic model whereas 26.18 m/s for the proposed model. It clearly shows the velocity was increased for the proposed model when comparing with basic model.



Graph 2: Velocity variation at different speeds for basic and proposed model

VIII. CONCLUSION

In the project work of modelling, outer styling with developed aerodynamics of existing intercity truck plying on Indian roads, a detailed computational analysis has been done. The Two prototype truck body has been modelled for performing numerical analysis using CFD software. Model No.1 is the existing/basic truck model and Model No.2 is that we altered and modified the existing/proposed truck model. Velocity given to the fluent analysis is 27.77 at a speed of 100 kmph. The resultant drag force of the baseline model/ basic truck model is 13136.58 N and the modified model we get is 10380.96 N. By these modifications the coefficient of drag is reduced by approximately 26% of basic one. In case of amount of fuel consumption for basic and proposed are likely to be 1.12% and 0.88% percentages are evolved. There is almost 27% of fuel saved for the proposed model. Finally on basis of above values obtained for both drag force as well as percentage consumption of fuel are most favourable to the proposed truck model when comparing with basic model. So we reduced the drag force, results in increased performance of the truck and reduced fuel requirement by conducting CFD analysis on ANSYS.

REFERENCES

[1] R. H. Heald, "Aerodynamic characteristics of automobile models," Bureau of Standards Journal of Research, vol. 11, Aug. 1933.
[2] S. Senger and S. D. R. Bhardwaj, "Aerodynamic design of F1 and normal cars and their effect on performance," International Review of Applied Engineering Research, vol. 4, no. 4, pp. 363–370, 2014.
[3] Ait Moussa, J. Fischer, and R. Yadav, "Aerodynamic drag reduction for a generic truck using geometrically optimized rear cabin bumps," Journal of Engineering, vol. 2015, Article ID 789475, 2015.
[4] R. Taherkhani, G. N. DeBoer, P. H. Gaskell, C. A. Gilkeson, R. W. Hewson, A. Keech, H. M. Thompson, and V. V. Toropov, "Aerodynamic drag reduction of emergency response vehicles," Advanced Automobile Engineering, vol. 4, no. 2, Article ID 1000122, 2015.
[5] J. Howell, M. Passmore, and S. Tuplin, "Aerodynamic drag reduction on a simple car like shape with rear upper body taper," SAE Int. Tech. Paper, Apr. 2013.
[6] Y. Zhang, Z. Zhang, S. Luo, and J. Tian, "Aerodynamic numerical simulation in the process of car styling," Applied Mechanics and Materials, vols. 16–19, pp. 862–865, 2009.
[7] S. H. Abdul Kareem, M. Al Obaidi, and L. C. Sun, "Calculation and optimization of aerodynamic drag of an open wheel car," Journal of Engineering Science and Technology – EURECA 2013 Special Issue, pp. 1–15, Aug. 2014.
[8] J. Abinesh and J. A. Kumar, "CFD analysis of aerodynamic drag reduction and improve fuel economy," International Journal of Mechanical Engineering and Robotics Research, vol. 3, no. 4, Oct. 2014.
[9] F. Mariani, C. Pogianni, F. Rishi, and L. Scappaticci, "Formula SAE racing car: Experimental and numerical analysis of the external aerodynamics," in Proc. 69th Conf. Italian Thermal Machines Engineering Association (ATI), 2014.
[10] Ansari and R. M. Mourya, "Drag force analysis of car by using low speed wind tunnel," International Journal of Engineering Research and Reviews, vol. 2, no. 4, pp. 144–149, Oct.–Dec. 2014.

[11] K. Nisugi, T. Hayase, and A. Shirai, "Fundamental study of aerodynamic reduction for vehicle with feedback flow control," JSME Int. J., Series B, vol. 47, no. 3, 2004.
[12] L. A. Raman and R. H. Hari, "Methods for reducing aerodynamic drag in vehicles and thus acquiring fuel economy," Journal of Advanced Engineering Research, vol. 3, no. 1, pp. 26–32, 2016.
[13] U. S. Rohatgi, V. S. Ko, and R. Pavlovs'ky, "Methods of reducing vehicle aerodynamic drag," in Proc. ASME Summer Heat Transfer Conf., Puerto, USA, Jul. 2012.
[14] S. M. R. Hassan, T. Islam, M. Ali, and M. Q. Islam, "Numerical study on aerodynamic drag reduction of racing cars," in Proc. 10th Int. Conf. Mechanical Engineering (ICME), 2013.