# CFD analysis of commercial bus models for improvement of aerodynamic performance

**N. Govindha Rasu*, A. M. Renil, S. J. Sachin, and J. kevin**
School of Mechanical Engineering, VIT University, Vellore - 632014, Tamil Nadu, India.
***Corresponding author: E-Mail: ngrasu@vit.ac.in**

**ABSTRACT**

In spite of extensive research and advancements in the automotive industry, fuel efficiency still continues to be a major problem. Though there have been many methods that come into the fore when addressing the issues related to fuel efficiency, cost effective methods are less. Where, aerodynamics of a vehicle like a car or a bus can be a parameter that may be a cost effective method. Thus by adopting proper physical modifications of the vehicles with easy manufacturability, we can create a model which has significant drag reduction.

Everyday millions of commuters use public transportation to travel to various places. There are different modes of transport available, like buses, trains, and cabs that transport people from one place to another. In present study bus is chosen, which is the mostly used as a public transport in the developing countries like India. In this study it is focused on fuel efficiency by reducing the aerodynamic drag coefficient of a bus model. Three bus models present in the current market are designed using ANSYS Design Modeler. Further the drag coefficient of these bus models are analyzed by using ANSYS Fluent. The drag coefficient of these buses is validated and benchmarks are set. Following the basic study, bus models with different aerodynamics are studied. Drag Coefficient of various bus models are compared and bus model with the least drag coefficient (Model-F) is proposed for the future use in the market.

**KEY WORDS:** Bus Model, Drag coefficient, Aerodynamics.

## 1. INTRODUCTION

Environmental issues are slowly becoming dominant in the current scenario of the automobile industry. Stringent norms are being imposed on pollution control and fuel efficiency. Focus of research and development in the industry is oriented towards not only the engine efficiency but also the aerodynamics of vehicles. Aerodynamics can significantly impact the fuel efficiency of an automobile. Aerodynamics is a branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with a solid object, such as an airplane wing. Aerodynamics is a sub-field of fluid dynamics and gas dynamics, and many aspects of aerodynamics theory are common to these fields. The term aerodynamics often used synonymously with gas dynamics, with the difference being that gas dynamics applies to the study of the motion of all gases, not limited to air. External Aerodynamics is usually associated with the flow around a solid closed body. This depends on multiple flow properties like flow speed, compressibility, viscosity etc. Namely two forces come into play when dealing with an aerodynamic study around a solid body. They are lift and drag forces.

Drag is the most significant force that occurs when air motion around a solid body is studied. Drag is important in a way that it directly opposes the direction of the body and a great amount of energy and power is required to overcome it, causing it to be a highly undesirable force. Automotive manufacturers try to factor in this force when designing automobiles. Drag force affecting a body in motion is usually calculated and represented by using a value called the drag coefficient (Cd). The drag coefficient has no unit. Forces that affect drag include the air pressure against the face of the object, the friction along the sides of the object and the relatively negative pressure, or suction, on the back of the object. Usually there are two types of drag that are commonly mentioned. They are pressure drag and skin friction. When a solid body in motion comes into contact with air particles, these particles tend to compress together at the frontal area of the solid body directly opposing the body's motion. This is called pressure drag. However these air particles tend to detach from the surface of the body and then rotate to finally aid the motion of the body. An airplane wing has one of the ideal structures that can make use of this effect of pressure drag. This can be further explained by the following figure.

Skin Friction or friction drag is the drag caused due to the friction between the solid body and the particles of the fluid that the body moves through. Friction drag is undesirable because it opposes the motion of the body and also causes wear and tear. Friction drag can be overcome by smoothening the surface of the body to such an extent that overall friction is reduced. But this is just a hypothetical solution as achieving smoothness at the smaller levels, even with the advancements present currently, is very difficult.

In our project, as the title suggests, we focused mainly on the aerodynamics of buses and how the drag could be reduced. Buses are one of the most common means of transport and also large consumers of fuel. Most of the power generated is used to overcome the aerodynamic forces that were mentioned earlier. So, by developing a model that can negate the aerodynamic we could be able to aid the struggle for fuel efficiency. Throughout our project, we have focused on bringing out an innovative design that can be used in the industry.

## 2. MATERIALS AND METHODS

**Mathematical modelling and grid generation:** The equations that govern steady compressible flow around the bus model includes, mass and momentum equations. Firstly, we studied bus models that are currently available in the market, along with models that were studied by papers that worked on this particular topic. After studying the particular models we decided on three models, one from the paper by Patil (2012). The second model was based on Caio Indusscar Apache an intercity bus model, similar to the SETC buses that are present in Tamil Nadu. The third model was based on the Volvo B9R buses present in South India. The models were created by using the ANSYS Design Modeler software, a feature by the ANSYS Inc. The three replicas were designed as follows.

While generating the mesh, sizing functions were used wherever necessary in order to obtain accurate drag parameters. The overall mesh of the body and the enclosure was given a medium mesh sizing with high smoothing and program controlled inflation. Grid generated is depicted in Fig. 1.

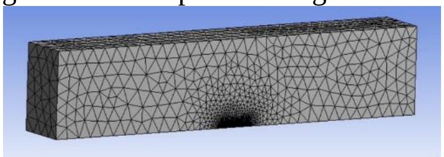


**Figure.1. Grid generated for the bus model**

**Boundary Condition:** The enclosure inlet plane was named velocity-inlet. Air coming through the inlet was given a velocity of 50 kmph, 75 kmph and 100 kmph which equates to 13.8 m/s, 20.83 m/s and 27.7 m/s. The road and the vehicle body were both made walls. The surrounding enclosure surfaces, being imaginary surfaces, were all named symmetry planes having a no slip condition. The outlet was named as pressure-outlet with its pressure set constant and equal to atmospheric pressure.

After successfully analysing the models we obtained the Cd values which will be shown later in the Results and Discussions section. The results obtained were validated. We then moved to designing new models and adding the geometrical additions and extensions. These models were analysed further until we could obtain a model with considerable drag reduction. This model is proposed at the end of the report. With this we aim to do a small part in aiding the fuel economy

The final solution was obtained by performing the iterations in two stages. With each progressive stage, the solver accuracy was raised by employing higher order equations. In the first stage, first order equations were used for the first 100 iterations so as to prevent the solution from diverging. Once sufficient convergence was achieved, the equation order was raised to second order for the rest of the iteration, so as to get a negligible and accurate drag coefficient.

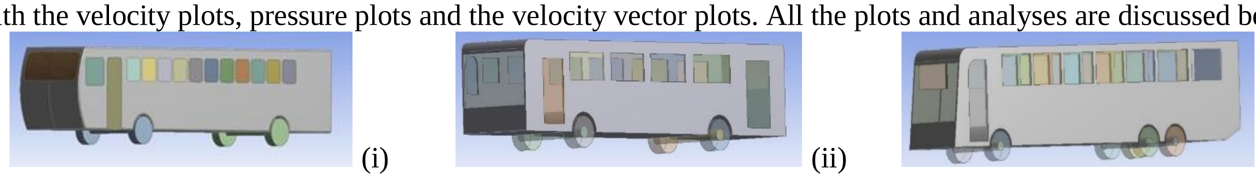
The code that we followed is the AIS-052 (Automotive Industry Standard). It is the Code of practice for bus body design and appearance. It is a standard which contains guidelines for the dimensions and appearance of a bus body. This standard must be adhered by all the bus body manufacturers. We have also adhered to this standard by following the guidelines of the length, width and wheel base.

The main constraint that we faced throughout this project is the problem of not being able to fabricate the models. This is impossible and would prove to be very expensive and unfeasible. So we had to address it by using the virtual wind tunnel that we made in ANSYS Fluent. ANSYS Fluent proved to be a very powerful tool that it has been used satisfactorily in all the papers that we have gone through in the duration of this project.

Other constraints that we have given in our analyses are the velocity constraints. As the buses under consideration are inter-city buses, they could touch high speeds provided they remain within the prescribed speed limits of various roadways. Therefore we have chosen speeds 50, 75 and 100 kmph which equates to 13.8 m/s 20.83 m/s and 27.7 m/s respectively, as the inlet velocities for our virtual wind tunnel. Apart from that, the frontal areas of the three models are 7.1 $m^2$, 8.2 $m^2$, 8.6 $m^2$.

## 3. RESULTS AND DISCUSSION

**Existing Bus Models:** Design of the bus models are modelled using ANSYS Dimension Modeler available in the ANSYS-15 version. The reference values and the limits of various parameters for the present analysis are fixed based on the existing bus models. Structural steel is the material assumed for the bus model, Tyre dimension and model used are 315/80 R22.5. The lower and upper limit in the length of bus model used is 12000 mm to 13700 mm. Existing bus models are depicted in Fig.2, and the dimensions of corresponding bus models considered for the present analysis is given in the Table-1. After the CFD analyses of the three base models, we obtained the Cd values, along with the velocity plots, pressure plots and the velocity vector plots. All the plots and analyses are discussed below.



(i) (ii) (iii)

**Figure 2. Existing bus models (i) Model-A (ii) Model-B and (iii) Model-C**

**Table.1. Dimensions of existing bus models**

| | Length (mm) | Width (mm) | Wheel base (mm) |
|---|---|---|---|
| Model-A | 12673 | 2522 | 6000 |
| Model-B | 12000 | 2500 | 6000 |
| Model-C | 13700 | 2600 | 6500 |

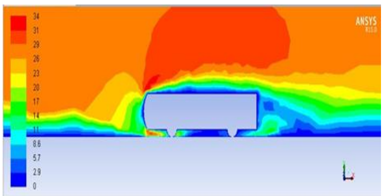


**Figure.3a. Velocity (m/s) Plot for Model -A**

**Figure.3b. Static Pressure (Pa) Plot for Model-A**

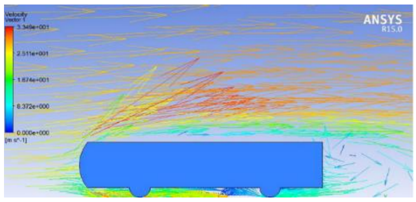


**Figure.3c. Velocity Vector (m/s) Plot for Model-A**

For the Model-A the velocity contour, static pressure and the velocity vectors are depicted in Fig. 3. Velocity contour shows that the velocity of the particles at the surface close to the body of the model is zero and increases as the particles move away from the body. The pressure plot shows that the pressure is highest at the frontal area further it detaches from the surface of the body. Further it is noticed the recirculation zone behind the vehicle. The drag coefficient ($C_d$) varies as 0.7481, 0.7392 and 0.7314 with increase in velocity of flow.

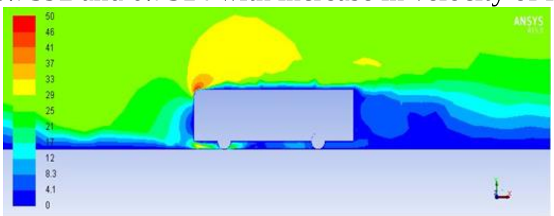


**Figure.4. Velocity (m/s) Plot for Mode-B**

For the Model-B the velocity contour, static pressure and the velocity vectors are depicted in Fig. 4. In this model it can be clearly seen that the square shaped frontal area acts as an obstacle, and thus drag in both the pressure plot and the velocity plot. The vector plot shows the recirculation. This is because of low numerical frontal area than model-A. For this model the drag coefficient varies from 0.6362, 0.6294, 0.6196 with increase in the flow velocity. For the Model-C the velocity contour, static pressure and the velocity vectors are depicted in Fig. 5. For model-C, because of the near streamlined shape there is less pressure and also an increasing recirculation energy which helps to reduce the drag. Further due to the frontal area of this model being the largest the $C_d$ value of this model is almost similar to model-B. The values are 0.6483, 0.6306 and 0.6235 with increase in the flow velocity.

After the analysis of the three base models, drag coefficient values obtained for the three velocities are given in Table-2. The present range of drag coefficient values compare very well with the values reported. With this it is decided to test the different designs of bus models for further reduction in drag coefficient.

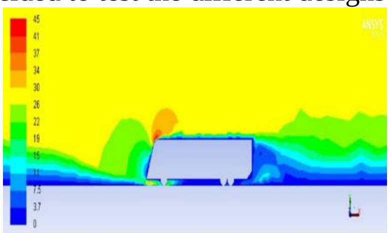


**Figure.5a. Velocity (m/s) Plot for Model-C**

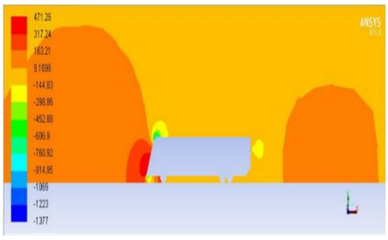


**Figure.5b. Static Pressure (Pa) Plot for Model-C**

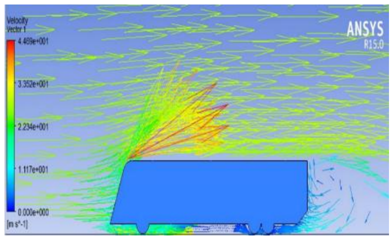


**Figure.5c. Velocity Vector (m/s) Plot for Model-C**

**Table.2. Drag coefficient values for different models**

| Bus Models | Velocity (Kmph) | Cd |
|---|---|---|
| Model-A | 50 | 0.7481 |
| | 75 | 0.7392 |
| | 100 | 0.7314 |
| Model-B | 50 | 0.6362 |
| | 75 | 0.6294 |
| | 100 | 0.6196 |
| Model-C | 50 | 0.6483 |
| | 75 | 0.6306 |
| | 100 | 0.6235 |

**Proposed Bus Models:** After the detailed analysis of three existing models, it is found that both the model-B and model-C have similar $C_d$ values and it was decided that the model-C be our base model for further studies. Also, Model-C was chosen because of its shape which can be easily modified to obtain the different models without much effort in geometrical additions and extensions. Further, three different models has been studied and it is named as models D, E and F as depicted in Fig.6.

(i) (ii) (iii)

**Figure.6. Proposed bus models (i) Model-D (ii) Model-E and (iii) Model-F**

Model-D has the same frontal features as model-C but had a tail plate at the rear roof. All the other features remain untouched. Model-E was made by filleting the top and bottom front edges of model-C. All the other features were kept intact. At the rear end however, we used the boat tail extension. No other additions were added. Model-F had a completely different shape altogether, aiming for a more streamlined shape at the front of the model. The rear end had features similar to model-C but spotted a tail plate at the rear roof. The rear part of this model is similar to model-E.

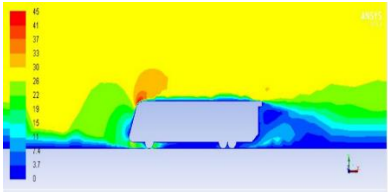


**Figure.7a. Velocity (m/s) Plot for Model-D**

**Figure.7b. Static Pressure (Pa) Plot for Model-D**

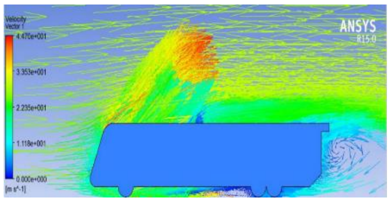


**Figure.7c. Velocity Vector (m/s) Plot for Model-D**

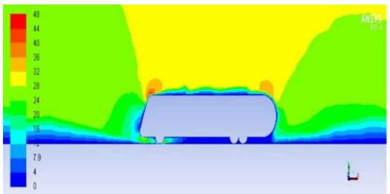


**Figure.8a. Velocity (m/s) Plot for Model-E**

**Figure.8b. Static Pressure (Pa) Plot for Model-E**

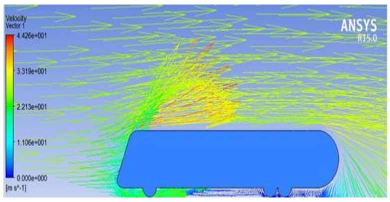


**Figure.8c. Velocity Vector (m/s) Plot for Model-E**

Based on the analyses of the three new models, drag coefficient values, along with the velocity contour plots, pressure plots and the velocity vector plots are brought out. The details for each case are discussed below. For the Model-D the velocity contour, static pressure and the velocity vectors are depicted in Fig.7. In this model the velocity plot clearly shows that on the front of vehicle top and bottom side of the bus model where the portion is curved allows more velocity flow which helps in bringing the drag down. In the pressure plot, the frontal area of the modal has high pressure and the rear end has low pressure and its difference determines the amount of drag. From the velocity vector plot, we can see that a vortex is created at end near the tail plate which helps in circulation and reduction in drag. The $C_d$ for the different flow velocity is 0.5427, 0.5415 and 0.5361.

For the Model-D the velocity contour, static pressure and the velocity vectors are depicted in Fig.8. In this model he velocity plot shows that the flow of air is more in the frontal top and rear area of the model which helps in drag reduction. The pressure plot shows that the pressure is high on the front side and low at the rear side of the model. Since we have given a boat tail extension on this model, the velocity vector plot does don't display a vortex on the rear side as compared to model-D, thus the recirculation is not carried out. Due to this $C_d$ of this is less compared to model-D. The values for the different flow velocity are 0.5819, 0.5791 and 0.5748.

For the Model-D the velocity contour, static pressure and the velocity vectors are depicted in Fig. 9. In this model the velocity plot shows that flow of air can pass through the frontal area smoothly as the front part is designed in a more aerodynamic friendly manner, and also high velocity is seen on the front top end of the model, The pressure plot shows high pressure at the front and low pressure at the rear end of the model. In velocity vector plot the tail plate contributes to recirculation and formation of vortex which reduces drag. The drag coefficient values for different inlet velocity values are 0.5011, 0.5001 and 0.4959. This model has low $C_d$ compared to other models. The $C_d$ values obtained from the analyses of the three new models are given in Table-3. So, Model-F is recommended for the future bus models with the reduction in the drag coefficient.

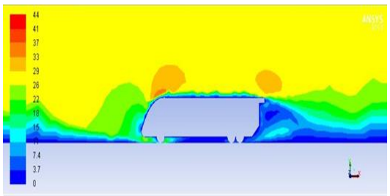


Figure.9a. Velocity (m/s) Plot for Model-F

Figure.9b. Static Pressure (Pa) Plot for Model-F

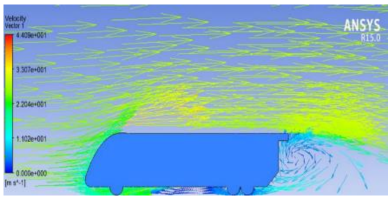


Figure.9c. Velocity Vector (m/s) Plot for Model-F

**Table.3. Drag coefficient values for proposed models**

| Bus Models | Velocity (Kmph) | $C_d$ |
|---|---|---|
| Model-D | 50 | 0.5427 |
| | 75 | 0.5415 |
| | 100 | 0.5361 |
| Model-E | 50 | 0.5819 |
| | 75 | 0.5791 |
| | 100 | 0.5748 |
| Model-F | 50 | 0.5011 |
| | 75 | 0.5001 |
| | 100 | 0.4959 |

## 4. CONCLUSION

After the detailed and parametric study a model is proposed with the lowest drag coefficient compared to presently available models. Also, the proposed model can be manufactured using existing manufacturing practices. When the drag coefficient values of the models-D, E and F are compared with those of the models-A, B and C, It is very clear that the newly made models perform much better in the study than the existing models. When comparing models-D, E and F, it's observed that models-D and F have much better $C_d$ values than model-E. This is because the boat tail extension used in model-E does not provide for air recirculation and hence the recirculation energy is negated. Further with model-F, the aerodynamically curve shaped gives less frontal area and hence reduces the drag coefficient value. Thus we propose model-F as the bus model that can be adopted by the current vehicle manufacturing agencies.

## 5. ACKNOWLEDGEMENTS

The authors would like to be obliged to VIT University, Vellore, INDIA for providing laboratory facilities and motivating for this project.

## REFERENCES

Abinesh J, Arunkumar J, CFD Analysis of Aerodynamic Drag Reduction and Improve Fuel Economy, International Journal of Mechanical Engineering and Robotics Research, 3, 2014, 213-215.

Amol Mangrulkar, Akshay Parab, Ammar Sakarwala, Bhushan Paste, Vaibhav Patil, Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5, International Journal on Recent Technologies in Mechanical and Electrical Engineering, 1, 2013, 7-13.

Naveen Kumar V, Lalit Narayan K, Narasimha Rao LNV, and Sri Ram Y, Investigation of Drag and Lift Forces over the Profile of Car with Rear spoiler using CFD, International Journal of Advances in Scientific Research, 2, 2015, 60-71.

Patil CN, Shashishekar KS, Balasubramanian AK and Subbaramaiah SV, Aerodynamic Study and drag coefficient optimization of passenger vehicle, International Journal of Engineering Research and Technology, 1, 2012, 60-71.

Sharma RB, Ram Bansal, CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction. IOSR Journal of Mechanical and Civil Engineering, 7, 2013, 28-35.