International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 7, September - 2012

Aerodynamic Study and drag coefficient optimization of passenger vehicle

C.N. Patil[*1], Dr. K.S. Shashishekar[2], A.K Balasubramanian[3], Dr. S.V.Subbaramaiah[4]

**1. Student, M.Tech, (Thermal Engineering), SIT, Tumkur, (Program Manager, CSM Software Pvt Ltd., Bangalore)*

2. Professor and Dean Academic, Dept. of Mechanical Engg, SIT, Tumkur

3. CTO, CSM Software Pvt Ltd., Bangalore

4. Technical Manager, CSM Software Pvt Ltd., Bangalore E

Abstract

Increasing demand for reducing fuel consumption and environment friendly vehicles is one of the major issues in automotive industry. Research on Aerodynamic design of Conventional BUS is desirable to see the possibility of drag reduction.

Aerodynamic flow simulation on one of conventional BUS is performed to demonstrate the possibility of improving the performance with benefits of Aerodynamic features around the BUS by reducing Cd which improves the fuel consumption. One of the Conventional BUS model is taken for optimisation and tried to reduce Cd by adding the external features along with front face modification. Optimisation of BUS is carried out by adding spoilers and panels at rear portion; this assessment has shown that drag can be decreased without altering the internal passenger space and by least investment.

Simple features installed and modifications performed at the rear end of the vehicle have shown capable of reattaching the flow along the BUS surface and reducing Aerodynamic Drag. Three simple low cost feature modifications at Rear end of the BUS have been performed. These technologies have shown around 20 to 30% of drag reduction of BUS.

1. Introduction

Environmental issues are becoming dominant in current scenario of automotive industry. Stringent norms are imposed on fuel efficiency and pollution control. Focus of Research and development in industry is oriented towards not only improving engine efficiency and also aerodynamics of vehicles. Shape of vehicle can influence to a large extent on drag of vehicle, which in turn contributes towards fuel consumption. The recent studies show that aerodynamic improvements of the tractor (Commercial Vehicles), passenger vehicles (BUS/CAR) are most important issues when it comes to fuel saving.

Figure 1 shows classical plot of the Drag and Resistance forces versus speed. It is evident from graph that aerodynamic drag force becomes dominant at higher speeds. For a vehicle running on typical highway at higher speeds, more than half of the engine power is required to overcome the drag. This led to study on aerodynamics is important for long-distance transport vehicles. It may become very true for vehicle running with speed between 60kmph and 90 kmph. In general, for a typical passenger vehicle at lower speed, the rolling resistance and aerodynamic resistance are equal. At highway speed of around 60kmph the aerodynamic drag is the main source of driving resistance, whereas the rolling resistance stays more or less constant. However, it should be mentioned that aerodynamic drag is still important below this speed.

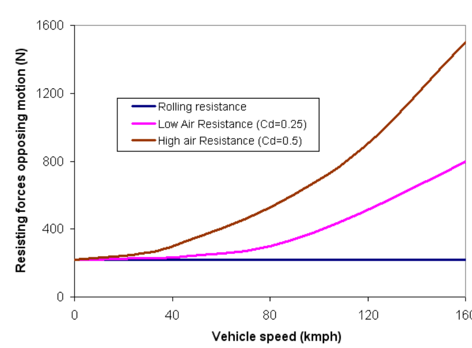


Fig. 1 vehicle speed v/s resistance forces

Various Numerical studies were carried out on Aerodynamics of vehicle mainly cars, Tractors and Trailers. Bahram Khalighi et al (2001) [1] have performed studies on simplified Square back model. They have implemented extended plates as modification at the rear end. They have achieved 6% drag reduction. Richard M. Wood et al (2003) [2] conducted experimental and computational investigations on Tractor-Trailer Trucks using simple low cost devices. In these studies, three simple low cost aerodynamic drag reduction devices have been developed. These technologies have shown a combined fuel savings of 10% at an average speed of 76 kmph. G. Frank et al.(2009) [3] worked on unsteady flow around Ahmed vehicle model using numerical methods.

2. Methodology

Present study involves investigation of aerodynamic characteristics and optimisation of conventional BUS using CFD methods. Aerodynamic study on Conventional BUS was carried out by modifying geometric futures at Rear end of the BUS.

CFD analysis is extensively used to optimize design from fluid dynamics point of view, before final prototype is made. This will shorten lead times and reduce cost of design. Complex behavior of the fluid flow can be captured in the CFD simulation studies with reasonable amount of accuracy. CFD solvers are matured in recent times to such an extent that the results obtained from analysis can be in good agreement with real world flow situations (qualitatively).

Analysis procedure and guidelines were standardized conducting aerodynamic flow studies around Ahmed model [3], for which experimental results was available along with Numerical calculations. Similar settings and methodologies were adopted for actual BUS to maintain the consistency and accuracy of results.

Conventional BUS used by Transport department shown in the Figure 2 was taken for optimisation; minimum dimensions of the base model are shown in the Figure 3. This model was named as Base Model, which represents majority of buses those have been running on roads at Highway speed. Base model behaves like a bluff body; modifications at front and rear end were carried out to reduce the drag coefficient.

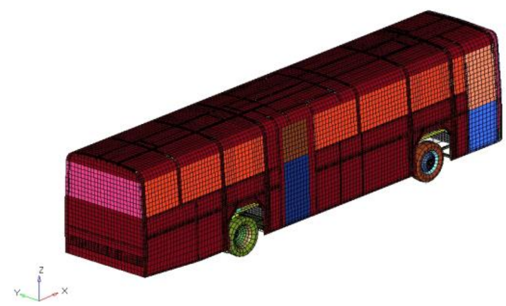


Figure 2: Surface Mesh generated on Base Model taken for optimization

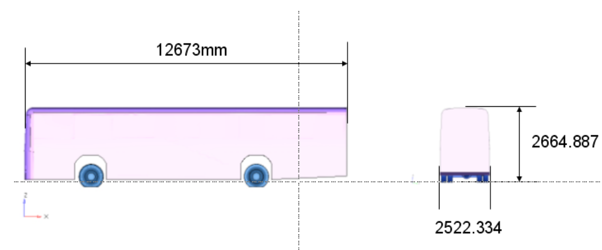


Figure 3: Base Model (Dimensions are in mm)

3-Dimensional flow (3D) analysis had been performed to understand the flow characteristics around a BUS body. Numerical studies were performed by solving the Navier Stokes equation with two equation turbulent model (k-epsilon).

Three modifications were tried in current studies as shown in Table 1.

First modification done in the front and rear end based on the results from base model. Modification at the rear end is tried out by complete covering the Rear portion of the vehicle with Boat like features as shown in the Figure 4. This modification is tried out without altering the side potion of the Base Model, to maintain the same space within the vehicle. This model was named as case no.1

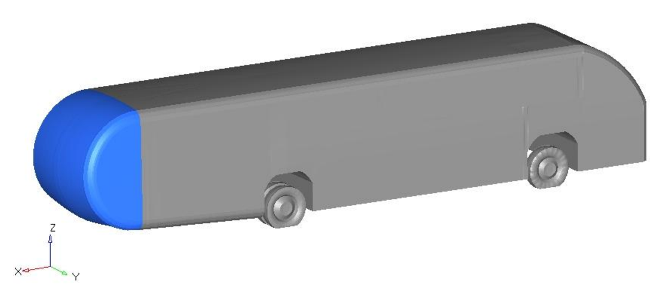


Figure 4: Geometry of Boat End Extension (Case1)

Based on literature study [2] cut features were implemented at rear end sides referred as panels. These panels were taken for optimized length, depth, angle and number based on previous studies referred, and is shown in the Figure 5. These modifications may facilitate mixing of low and high momentum layers coming from different sides and improving pressure recovery in the wake. Further on this modification is referred as Case no.2.

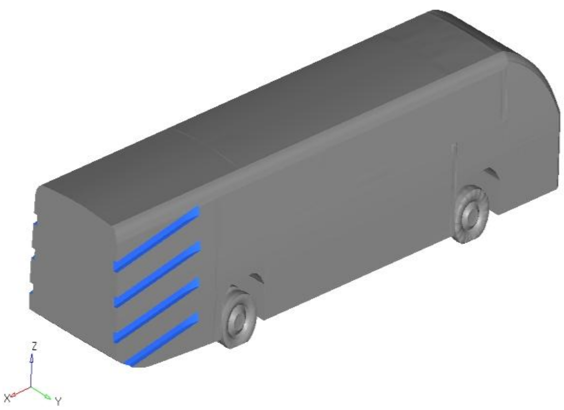


Figure 5: Panels (cut features) added at rear end

Normally spoilers are designed for vehicle to reduce aerodynamic drag, to create localized force and aesthetics. Spoiler will be simplest type of modification as they can be attached to current models on road also. Base model showed large wake with very low velocities at back. This could be remedied by attaching an airfoil wing at the rear top of the Base model as shown in the Figure 6. Further on this modification is referred as Case no.3

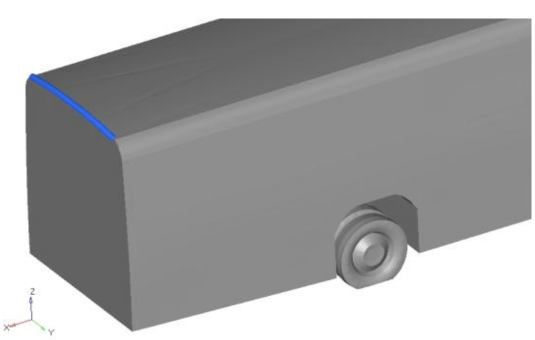


Figure 6: additions of Spoilers at Rear End (Case3)

All cases for which modifications were tried at rear end, which may influence Drag reduction, were shown in Table 1.

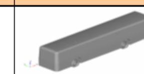
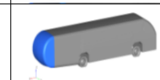
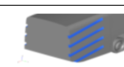
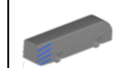
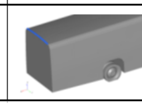
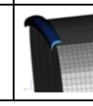
Case No	Case Name	Features	
	Base Model		
1	Boat End Extension		
2	Panels added at Rear End		
3	Spoiler at upper side of Rear End		

Table 1: Case Definition

3. Result and Discussion

Computational results for different models as discussed in previous section were presented here, starting with base model. Velocity magnitude and vector plots for base model are shown in figure 7, figure 8 and figure 9 respectively. It is observed that there is a large stagnation zone in front of the bus. Also big recirculation with dead zones is seen at the back of the vehicle. Wake velocity pattern was shown in section plots along the length direction in figure 10.

It can be seen that the sharp edges in the front portion of bus produces strong negative pressures as shown in figure 11.

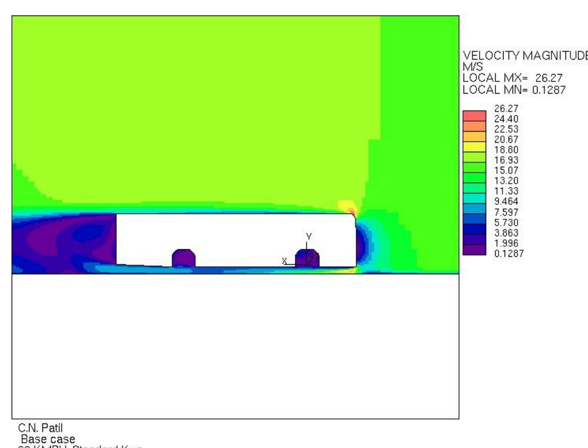


Figure 7: Velocity plots at mid section of Base Model

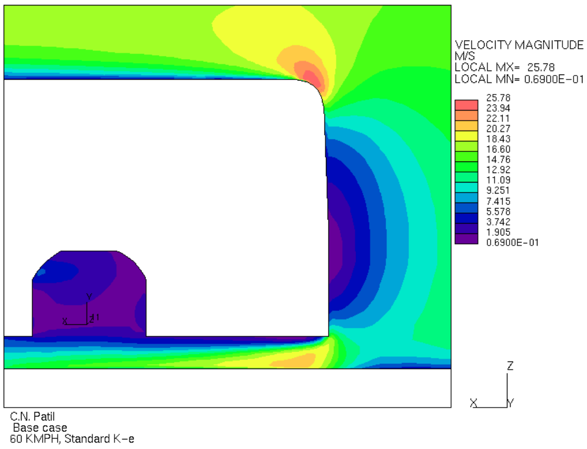


Figure 8: Velocity plots at Front portion of Base Model

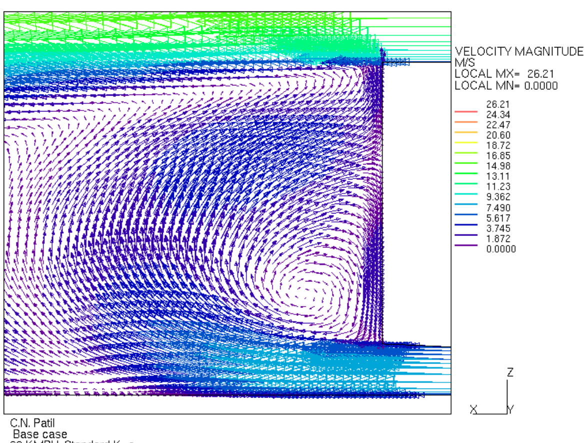


Figure 9: Velocity plots at rear portion of Base Model

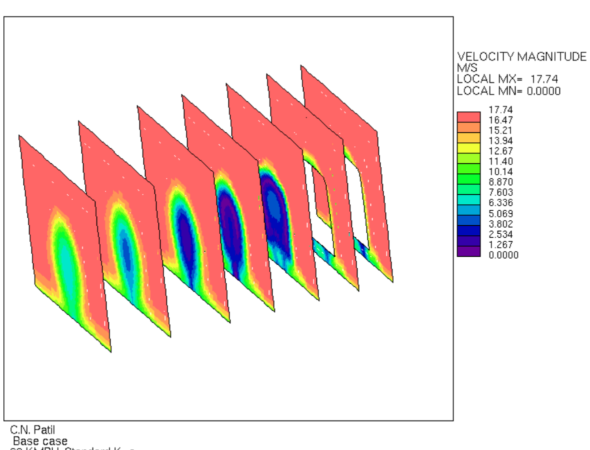


Figure 10: Velocity plots at rear portion of Base Model at various sections

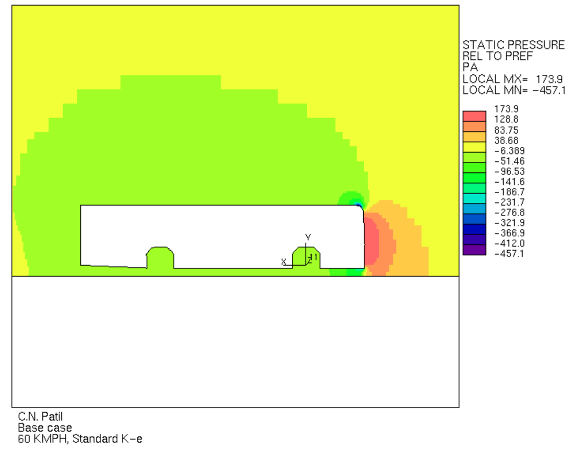


Figure 11: Pressure plot at mid section of Base Model

The adverse pressure gradients on side walls of Base Model leads to flow separation or low velocity flow along sides. This in turn creates large dead zones at rear end of Base Model as shown in Figures 9. Dead zones at the rear end of vehicle are seen clearly in velocity plots shown in Figure 8 and 9. Low pressure zones are noticed at the rear end of vehicle as shown in pressure plots in Figures 11, 12 and 13. Drag coefficient achieved was around 0.53.

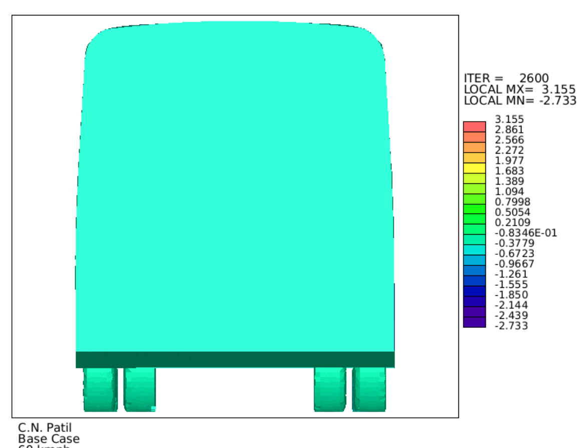


Figure 12: C_p plot at Rear End of Base Model

ISSN: 2278-0181
Vol. 1 Issue 7, September - 2012

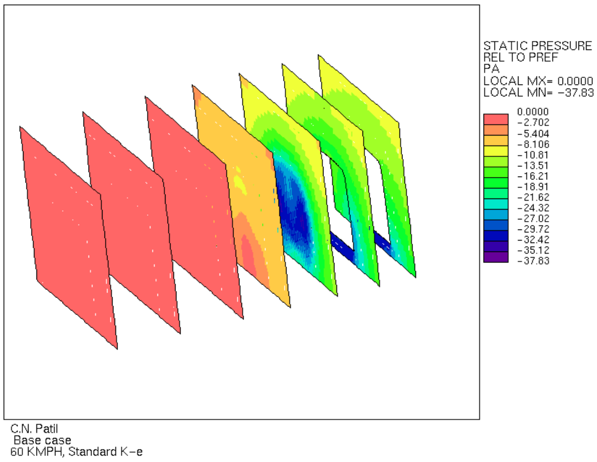


Figure13: Static pressure plot of Base Case at Rear end

It is observed that there is an increase in pressure along side walls of the Base Model due to Sharp corners. As a first modification rounded corners are incorporated at front edges. This has reduced considerably stagnation zones in front of BUS as shown in the Figure 14. This may be due to favourable pressure gradients produced by rounded corners.

Along with rounded corners in the front, Boat like modifications at rear end was tried out in Case No.1.

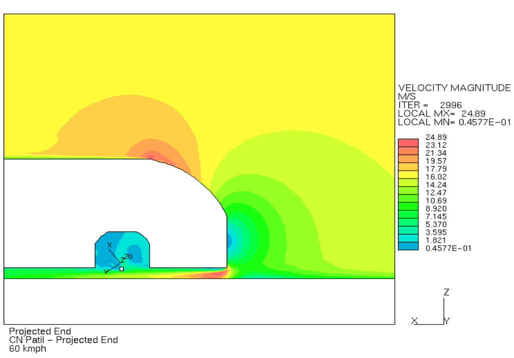


Figure 14: Velocity plots at Front portion of the modified model

Velocity comparison plots case no 1 and base case was shown figure 15. It can be seen that dead zones at rear end is reduced to lot extent in lateral section when compared to base case. Cp and Static pressure were show in figure 16 and 17. It can be seen that static pressure recovery is good at back of the vehicle. As seen in C_p plots, uniform pressure is not maintained at rear end, this may be due to the interaction of the high momentum flow coming from the top and bottom of the vehicle with the low momentum flow from sides. Drag coefficient for this model is around 0.49. Reduction in drag was 6.57% when compared to base case.

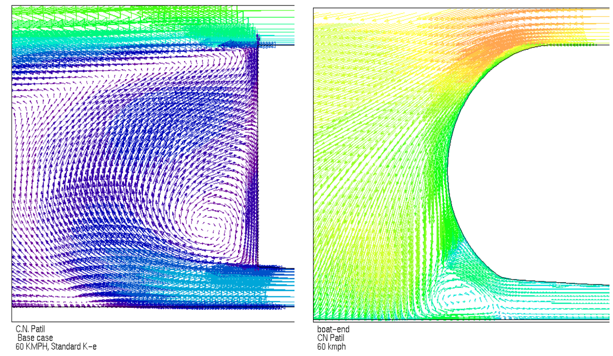


Figure 15: Velocity plots of Base model v/s Case No.1

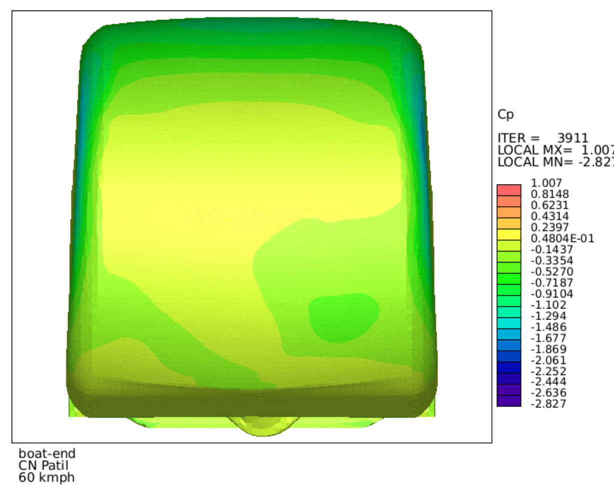


Figure16: C_p plot at Rear portion of Boat End model (Case No.1)

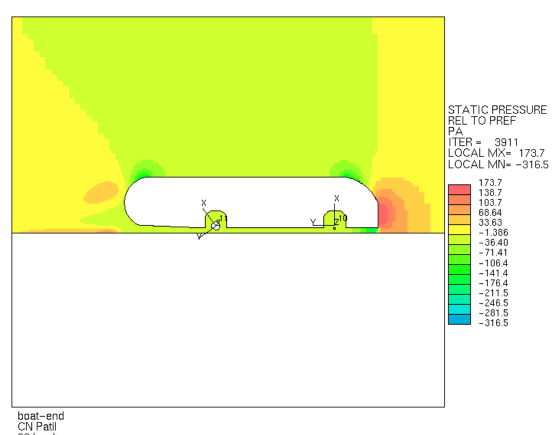


Figure17: Pressure plot at mid section of Boat End model (Case No.1)

It was found from literature [2] that slots provided at rear end (Panels) will help to improve the performance of bluff bodies. This type of modification tried in case no. 2.

Over all velocity distribution for this case was shown in figure 18. It can be seen from this figure that low velocity zone over top of the bus is completely vanished. Comparative flow picture with base case at rear end was plotted in figure 19. Flow from sides is mixing well with high momentum flow from top and bottom. This has directly led to reduced low velocity wake.

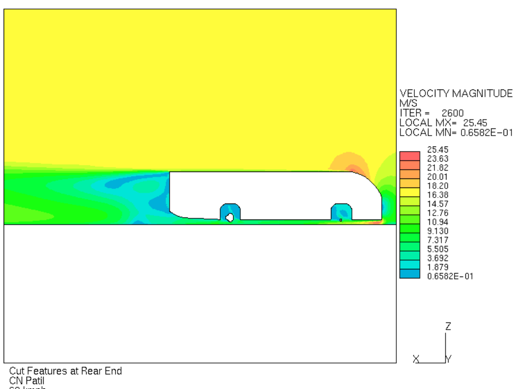


Figure 18: Velocity plots at mid section of Case No.2

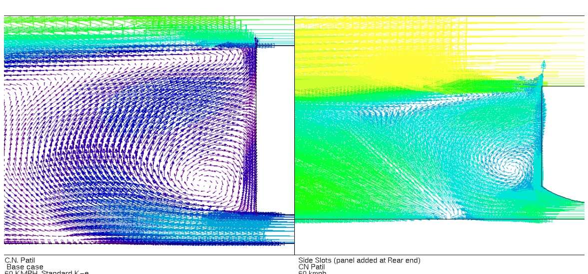


Figure19: Velocity plots of Base model v/s Case No.2

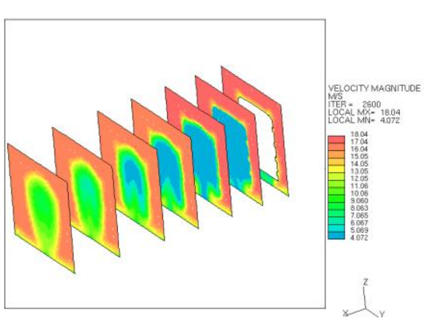


Figure 20: Velocity plots of Case No.2 at various Cross sections

Velocity map at various sections at rear end of the vehicle was shown in figure 20. It is evident from this plot that dead zones are reduced to a great extent when compared to base case.

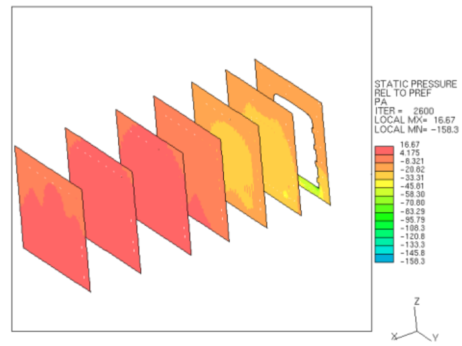


Figure 21: Static pressure plot of Case No.2 (panel added at Rear end)

Static pressure at various section in wake for this case was shown in figure 21, it was clearly seen that pressure recovery at the back side was improved greatly. This might have helped to reduce drag. Cp plot at rear of the bus was captured in Figure 22. With this modification drag coefficient value computed was around 0.39. It was around 25.82% reduction in drag when compared to base model.

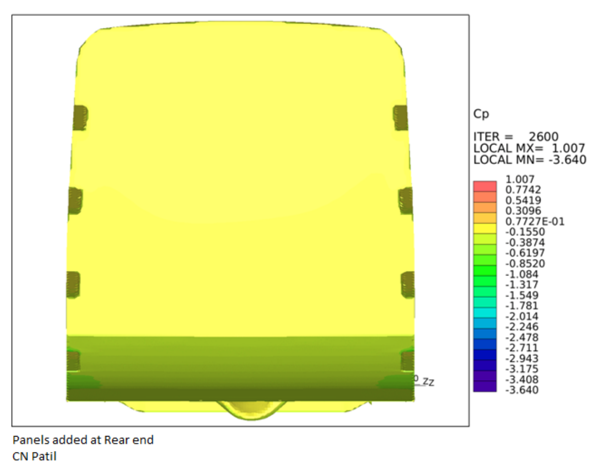


Figure 22: C_p plot at Rear portion of Case No.2 (panel added at rear end)

One more modification is tried out using Spoilers added at rear end of the BUS of base Model in Case No. 3. Velocity distribution for full vehicle, rear portion at middle lateral section and wake section were shown in Figures 23, 24, and 25. It is evident from figure 23 that wake has different mixing zones. Addition of this spoiler has forced the part of flow toward back surface of the vehicle and also into wake at rear as seen in the figure 24. Uniform flow pattern were observed at various section in the wake (figure 25). It is possible that this type of pattern was achieved, may be due to mixing of high velocity flow which directed from top.

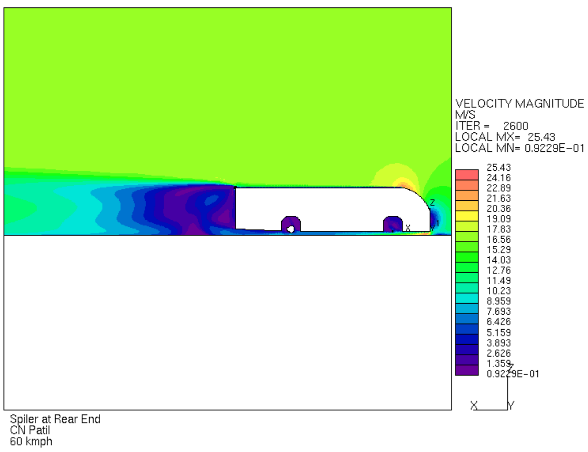


Figure 23: Velocity plots at mid section of Case No.3

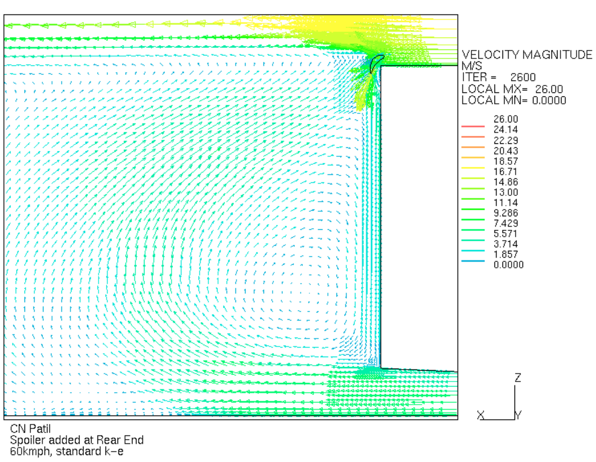


Figure 24: Velocity plots at rear portion of Case No.3 (spoiler added at rear top)

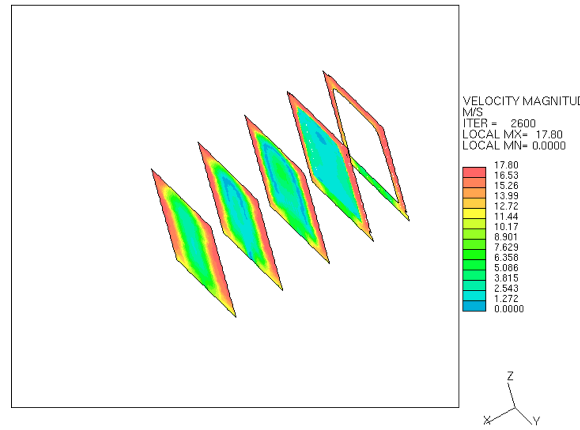


Figure 25: Velocity plots of Case No.3 at various Cross sections

A strong current high velocity flow is directed along rear end of the BUS, which intern reducing the low momentum flow. This has produced good pressure recovery as seen in Figures 26 and 27. It has reduced dead zones compared to base case.

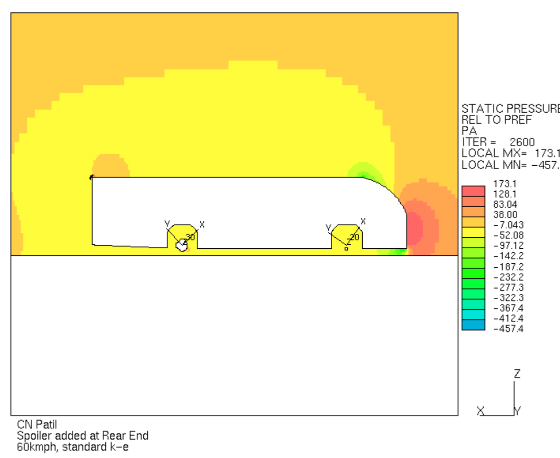


Figure 26: Pressure plots of Case No.3 at mid section (spoiler added)

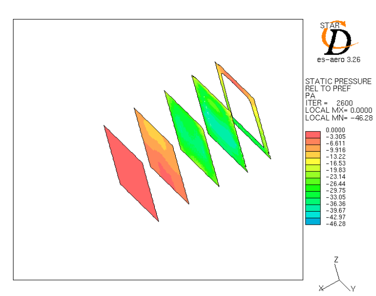


Figure 27: Static pressure plots of Case No.3 at various Cross sections

It was noticed from Figure 24 that small recirculation zone present at bottom side, this can be addressed by incorporating additional modifications at bottom side of Rear end. Coefficient of pressure was plotted in figure 28 at back face. It was clearly seen that uniform pressure distribution was observed. This was directly responsible for reduction in drag coefficient. The C_d was around 0.40. It looks that this is slightly higher case no.2. This may be due to the bottom recirculation zone. The drag reduction was around 24.42% compared to the Base Model.

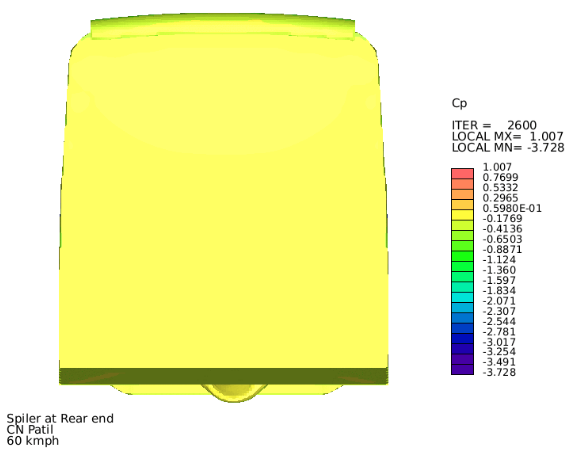


Figure28: C_p plot at Rear portion of Case No.3 (spoiler added at rear end)

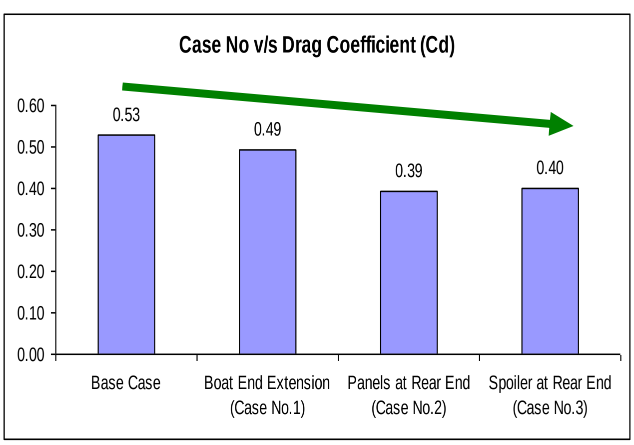


Figure 29: Drag coefficients of all cases from numerical result

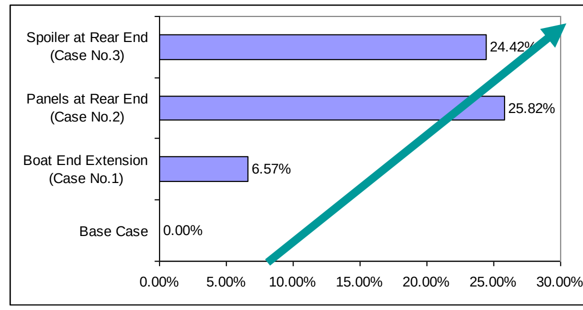


Figure 30: Drag in % with respect to Base Model of all cases from Numerical result

The simulations were quantitatively summarised in figures 29 and 30. Various changes implemented in Case1 (Boat end extension) Case 2 (Panels added at rear end) and Case 3 (spoiler added at Rear end) shows considerable reduction in drag. It was observed that Drag coefficients of Base Case, Case No.1, Case No.2, and Case No.3 are 0.53, 0.49, 0.39, and 0.40 correspondingly. Comparative reduction in drag was plotted in figure 30. It was observed that 6.57%, 25.82%, 24.42% of Drag reduction achieved for Case 1, Case2, and Case3 when compared to base model.

4. CONCLUSIONS

This study demonstrates the possibility of improving the Aerodynamic performance by different geometrical features around a passenger BUS. These features contributed towards reduction of C_d which impacts fuel consumption. Further improvements can be possible by combining any of geometrical features discussed in this study.

5. REFERENCES

[1]. Bahram Khalighi, S. Zhang, and C. Koromilas (2001), “Experimental and Computational Study of Unsteady Wake Flow behind a Bluff Body with a Drag Reduction Device” 2001 Society of Automotive Engineers, 2001.

[2]. Richard M. Wood, and Steven X. S. Bauer (2003), “Simple and Low-Cost Aerodynamic Drag Reduction Devices for Tractor-Trailer Trucks” SAE International, 2003.

[3]. G. FRANCK, N. NIGRO, M. STORTI, J. D’ELÍA (2009), “Numerical Simulation of the flow around the Ahmed vehicle model” Latin American Applied Research 2009.

[4]. Masaru KOIKE, Tsunehisa NAGAYOSHI, Naoki HAMAMOTO, (2004), “Research on Aerodynamic Drag Reduction by Vortex Generators” Mitsubishi Motors Technical papers 2004.

[5]. H. Lienhart, C. Stoots, S. Becker "Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model)" Lehrstuhl für Strömungsmechanik (LSTM), Universität Erlangen-Nürnberg, Cauerstr. 4, 91058 Erlangen, Germany