Abstract
The objective of this study is developing methodology of external aerodynamic analysis and simulation analysis of a passenger BUS ISUZU SAZ HC 40 for prediction of drag coefficient and its impact on fuel consumption with experimental validation. It has been checked that to resolve the air drag, the automobile absorbs possible the necessary engine power. Baseline model of passenger BUS ISUZU SAZ HC 40 was evaluated for estimating fuel consumption using Computational Fluid dynamics (CFD) methodology. The effects of the CFD were confirmed with the variability of experimental data. ISUZU architecture has been designed to reduce fuel usage with criteria such as window tilt, rounding and tapering corners and rear draft angle. Optimized ISUZU design is also ensured to meet functional specifications as Uzbek standartation. At Turin Polytechnic University in Tashkent, the prototype of the integrated ISUZU system was experimentally evaluated. The CFD and experimental test findings were reviewed and considered to be in good agreement. With the streamlined configuration of the ISUZU, it is observed that while the weight of the ISUZU was raised, the fuel efficiency remained almost the same.

Key words: Computational Fluid dynamics (CFD), Aerodynamic Optimization, Flow Rate.

Introduction
The increase of fossil fuel depletion is one of the world’s most important dilemmas in the coming decades. Petroleum products constitute 40% of the overall fossil fuel volume. Diesel and fuel are the main items of petroleum that are disintegrated day by day. 66 percent of the total petroleum products consumed by different sectors are not consumed for transport. ISUZU is Uzbekistan’s largest mass transportation system. For the last 10 years, Uzbekistan highways have improved significantly, and travel time has been shortened as they can operate at high speeds. Consequently, engendering ISUZU with better fuel efficiency has become critical. Strict government legislation and increasing fuel prices pressured vehicle manufacturers to make ISUZU fuel efficient. Uzbekistan’s ISUZU body manufacturers are concerned only with ISUZU’s aesthetic sense. Aerodynamics and its effect on fuel consumption of high-speed moving ISUZU are given less importance. The engine power is used in a moving vehicle to resolve tractive resistance, which is the mixture of rolling resistance and aerodynamic. The rolling resistance at lower speeds should exceed the aerodynamic resistance. But as the vehicle speed rises, the rolling resistance becomes overcome by the aerodynamic resistance. The pressure on the engine, which in effect improves the fuel efficiency of the ISUZU, is thus minimized by rising the aerodynamic resistance at high speed.

Aerodynamic analysis and optimization of wind deflector in ISUZU.

The amount of energy needed to overcome aerodynamic drag is determined by the vehicle’s efficient frontal surface area, vehicle form, and vehicle speed. Aerodynamics of ISUZU is affected by the shape of the ISUZU cab, the transition between the back of the cab and the front of the ISUZU, the gap distance between the ISUZU, the underbody of the truck, and the rear edge of the ISUZU. Principles that reduce the effort of aerodynamic drag are: the cab should be shaped as aerodynamically as possible and a smooth transition should take place between the top of the cab and the top of the ISUZU (through height matching or connections that create a smooth line between the two), the gap distance ISUZU should be minimized, the two should be positioned...
as close to each other as possible or the two should be positioned as close as possible. The model is based on ISUZU and built using CAD software. This model was subjected to external flow of 60kmph, its initial drag obtained and flow around the body analyzed.

Figure 1: Side and top view of passenger Bus ISUZU SAZ HC 40.

The program used is Solid functions for both producing the CAD model and carrying out the aerodynamic research. Thanks to its ease of use and responsive GUI, this program was picked. The model cad was produced using the defined measurements. Then the workbench for flow simulation was installed. The initial parameters are set; namely the fluid velocity of 16.67 m / s in –z direction, the fluid is selected as air and the flow is said to be laminar and turbulent. As the angle of the wind deflector rises streamlines avoid hitting the ISUZU and the region of the ISUZU in direct contact with streamline decreases resulting in lower resistance. CFD modeling is widely used before the final version is produced to refine functionality from a fluid dynamics viewpoint. This will reduce lead times and design costs. Complex conduct of the fluid flow can be described with reasonable accuracy in the CFD modeling experiments. In recent times, CFD solvers have evolved to such a degree that the results obtained from the study can be in good agreement (qualitative) with real world flow scenarios. Research protocol and instructions for performing aerodynamic flow experiments were developed for which experimental results and numerical measurements were possible. To ensure the quality and precision of the tests, identical configurations and methodologies have been implemented for the real ISUZU.

Result and Discussion

Computational findings for various models were described here, beginning with the base model, as mentioned in the previous section. Figure 2 display the velocity magnitude and vector plots for the base model. It is found that in front of the ISUZU there is a wide stagnation region. On the back of the vehicle is also seen large recirculation with dead zones. In section plots along the length direction in figure, the trend of wake velocity was shown. It can be seen that as shown in figure 11, the sharp edges in the front of the ISUZU produce strong negative pressures. Base Model’s unfavorable pressure gradients on side walls result in division of flow or low velocity flow between sides. This in turn creates large dead areas at Base Model’s rear end as shown in Figures. Dead zones at the vehicle’s rear end are clearly seen in the speed plots shown in. As shown in the pressure plots in Figures 2-3 low pressure areas are observed at the rear end of the car. The resultant drag coefficient was about 0.53.
Aerodynamic flow simulation is conducted on one of the traditional ISUZU to show the possibility of improving performance with the advantages of Aerodynamic features around the ISUZU by -Cd which improves fuel consumption. One of the traditional ISUZU layout is taken for optimization and attempted to minimize Cd by incorporating the exterior character-
istics and alteration of the front side. ISUZU enhancement is achieved by installing spoilers and panels in the rear portion; this assessment has shown that drag can be minimized by altering the internal passenger space and by reducing expenditure. Easy software mounted and improvements at the rear end of the ISUZU showed the ability to re-attach the flow along the surface of the ISUZU and minimize Aerodynamic Drag. Three basic changes to the low cost feature were produced at the rear end of the ISUZU. Such innovations demonstrated around 20-30% reduction in ISUZU drag. This study demonstrates the possibility of improving the Aerodynamic performance by different geometrical features around a passenger ISUZU. These features contributed towards reduction of $C_d$ which impacts fuel consumption. Simulations were run at zero yaw angle. Generally, a solution is assumed converged when the residuals become constant and do not change with additional iterations.

![Drag monitor plot](image1)

![Velocity vector plot](image2)

Figure 4. a) Drag monitor plot b) Velocity vector at the rear region.

As we did not consider wheel rotation that’s why actual value of $C_d$ of this ISUZU will be slightly higher approximately 10-15% more. This is one of the reasons of getting small drag coefficient value. As we have mentioned earlier that the narrower the wake the smaller the drag. Air gets stagnated at very thin line at the front, which results comparatively small total pressure at front that yields consequently lower drag force. Drag coefficient of this redesigned exterior body is found to be much lower due to the combined effect of narrow wake region and less frontal stagnation area. Less drag force means less fuel burning at highway speed so environment friendly and economical. Curving the front and rear end of the ISUZU may cost few passengers but it should not yield any profit loss in the long run.

References
