RESEARCH ARTICLE OPEN ACCESS

Investigation To Reduce Drag of a Noah 2020 Car by Using Contour Bump

Abdullah Al Hasan [1], Md. Shamim Rayhan [2]

[1] Faculty, Department of Mechanical Engineering, City University, Savar, Dhaka

[2] Faculty, Department of Mechanical Engineering, City University, Savar, Dhaka

ABSTRACT

NOAH car is a four wheeler minivan or passenger car which is a popular means of transportation in Bangladesh due to its comfortability, provision to go anywhere like for any occasion, family program and official use. But the aerodynamic drag has a great contribution on its energy and fuel consumption which is 0.4001 for the base model. In our investigation we used ANSYS Fluent software with CFD simulations for add-on devices. We used contour bump on the rear roof of the existing model without changing its external and internal design. The 3D designs of solid model are taken into account for the simulation of flow visualization around the car. Coefficient of drag CD was found numerically for all the models where we got 0.3859 for using one bump which reduce the drag by 3.5% and the maximum reduction was 0.3843 by 3.95% for two bumps configuration. Other six configurations are also effective but less efficient than the two proposed model of one and two bump configurations. Also the fuel consumption was reduced by maximum of 2.37% for two bump and 2.1% for one bump configuration. Also we have got an effective result for stream wise configuration of two bumps where we got maximum drag reduction of 3.85% which reduce the fuel consumption by 2.31%.

Keywords: - **NOAH** 2020, Drag Force, Co-efficient of Drag, Fuel Consumption, Drag Reduction.

I. INTRODUCTION

As the fuel cost has improved drastically, we tried to reduce the fuel consumption of this car by reducing its drag coefficient. So to do this, we use contour bumps on the rear roof of this car but did not change its existing model. Then contour bump has been considered as the drag reduction device in our investigation which is made in aerodynamic shape. With this we tried to save the fuel cost of this popular vehicle as well as to improve our national economy.

Aerodynamic is a process of fluid or air flow around an object or an object moving through air. Airplane fly on the basis of aerodynamics law. Besides this rocket, jet plane as well as ground vehicles like bus, truck, car etc are also affected by aerodynamics. It is a branch of fluid dynamics. It can be divided into external and internal categories. Our research is basically on external category which deals with the flow around a solid objects and related with generic geometry models, contour plots, vector plots, XY plots etc

II. LITERATURE REVIEW

So many studies were done on the reduction of drag and fuel consumption of different vehicles by using different methods. Some of them got a drastic change in percentage of drag reduction some got different issues and suggested further improvement. In this chapter we will discuss about some of the research related to the drag reduction of different vehicles and how those contributed to balance the fuel economy.

A. Vortex Generator

Vortex Generator in different heights can cause increase and decrease in drag. It was suggested to install a delta-wing shaped VG at an angle of 15^0 against the vehicle centre line [1]. Also a studied was carried out of 3D-bluff body with non-conventional geometry of VGs by passive flow control experiment [2]. They found a maximum of 12% reduction in drag with a set-up of PIV to identify 3D structures of the flow [2]. Also another investigation was done further for three configuration of VGs along the width and found a maximum of 14% reduction in drag for an active mechanical VGs configuration which was motorized and remote controlled from outside [2].

B. Diffuser

An actively translating rear diffuser device under the rear bumper can be used to maintain the streamlined automobile's under body configuration to reduce aerodynamic drag of a passenger car [3]. This devices was only active when the vehicles is in high speed resulting in an increase of the base pressure of the vehicles body. Seven types of diffuser was investigated for 130 km/h and the drag was reduced by 4.78% [3]. Also to reduce fuel consumption of both sedan and wagon type vehicles, a underbody diffuser was added and was investigated by CFD method which showed a 12% reduction in drag for a sedan car with 8^0 diffuser angle and a 3% reduction for wagon type with 5^0 diffuser angle [4].

C. Add-on Devices

The mechanisms of geometrically optimized bumps on the rear end of the cabin roof of a generic truck was a promising strategy for reducing drag. These device was expected to reduce the drag by deflecting the flow behind the cabin back slightly downward but in some cases authors suspected some additional drag which might raise the overall drag and found the maximum of 6-10% reduction in drag for different configurations [5].

A CFD simulation was done over a fast back passenger car by adding devices like roof cover, spikes at the rear end of roof [6] and noticed a slightly higher pressure at the front wheels than rear creating drag but a lower drag co-efficient than a real life drag of passenger car.

D. Rear Spoiler

Rear spoiler on trunk to reduce drag of a HONDA city using oil flow visualization techniques using 30 m/s free stream velocity and smoke visualization of 10 m/s where a 5-14% difference in pressure coefficient [7].

Spoilers were set on different angles of 10^0, 15^0 and 20^0 to get effective reduction in drag using k-epsilon method, where 47.7 m/s for 20^0 and 43.3 m/s for 10^0 spoiler [8].

III. MODEL PREPARATION

The generic model of vehicle with relevant dimensions are shown in **Fig.0.1**. The overall length of the car is 1065mm, width is 314mm, height 370mm, wheel width 65mm, front grille height is 178.25mm.

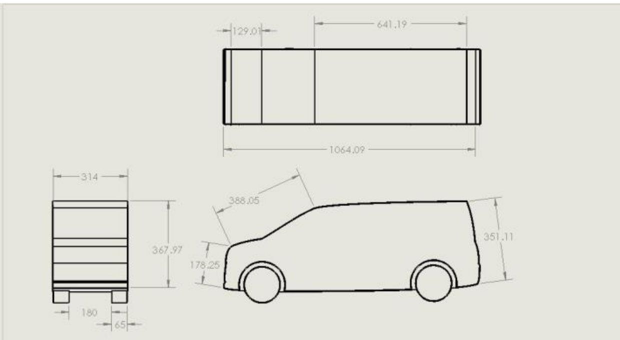


Fig.0.1 NOAH 2020 generic model with dimensions.

A. Bump Model With Dimensions

As the upper surface of the car was not flat so designed the bump over a plane surface. We used different bumps of similar dimensions in a symmetric distance from one another in span wise direction. The model and the different dimensions of the bump are shown in **Fig. 0.2**.

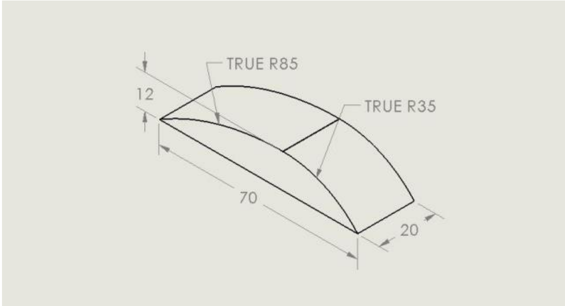


Fig. 0.2 Contour bump model and dimensions.

IV. NUMERICAL SIMULATION

CFD analysis was performed to investigate the flow analysis around the base model and the suggested model with add-on devices. The domain velocity inlet was 2 times of overall length and the pressure outlet was 6 times of the overall length shown in

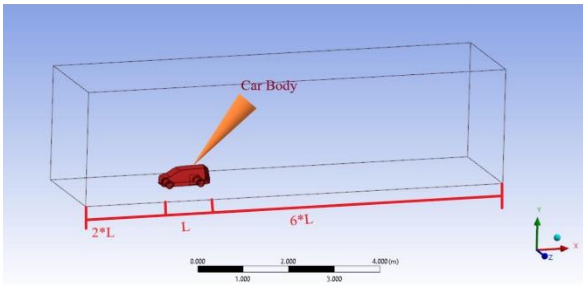


Fig.0.3 Enclosure orientation around the car model.

TABLE I
PHYSICAL PARAMETERS OF SOLVER SETTING

Physical Properties	**Parameters**
Flow characteristics	Steady, stationary, incompressible, turbulent, RANS
CFD Simulation	3D
Solver Type	Pressure based
Time	Steady
Turbulence Model	k -epsilon (2 eqn.)
Model	Realizable
Near-wall treatment	Non-equilibrium wall function
Inlet Velocity	30 m/s
Pressure Outlet	1 atm
Solution Methods	
Pressure Velocity Coupling	SIMPLE Scheme
Pressure	Standard
Momentum	Second Order Upwind
Turbulent kinetic energy	Second Order Upwind

The solver settings includes the viscous model, reference values, boundary conditions and solution control etc which are shown in TABLE II.

A. Computational Meshing

ANSYS FLUENT CFD is one of the mesh generation software, which provides high quality mesh for numerical simulation. The geometric models have done using the tetrahedral meshing elements that are easy to fit in small corners and are efficient enough to give accurate results. The meshing grids were divided into coarse, medium and fine meshing grids and the optimum number of meshing elements that have been achieved during the numerical analysis and using the available ANSYS FLUENT software in below TABLE IIIIV and Fig 0.4 for span wise directed bumps and Fig 0.5

For stream wise directed bumps.

TABLE VVI
MESHING METHODS

Mesh Default Settings	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Sizing	
Size Function	Proximity and Curvature
Max Face Size	Default (0.515130 m)
Growth Rate	Default (1.20)
Quality	
Target Skewness	Default (0.900000)
Smoothing	High
Inflation Option	Smooth Transition
Transition Ratio	0.272

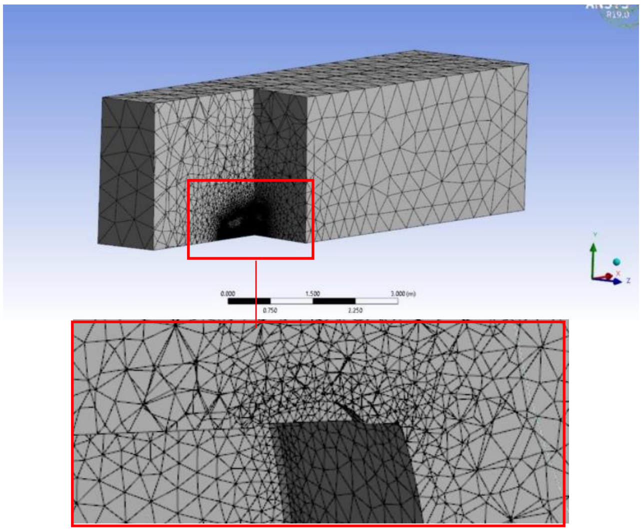


Fig 0.4 Mesh characteristics over the bump and the car body.

The coarse mesh has created with smooth inflation layer over the bump and the perfect elements of tetrahedral meshing. It indicates the smooth flow of air over the bump and the flow around the car body.

1) Stream wise Mesh:

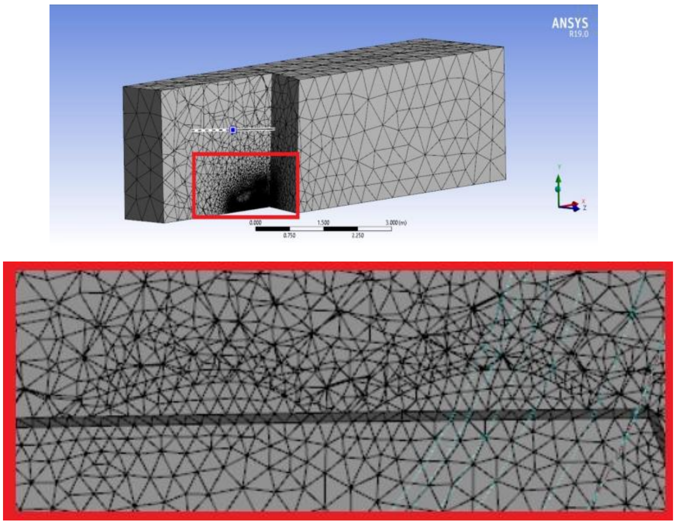


Fig 0.5 Mesh characteristics for stream wise bumps.

B. Residual and Drag co-efficient graph

1) ***Residual Characteristics in Span Wise Direction:*** The simulation has done following the boundary conditions for 1000 iterations. The residual graph was converged and it took 110 to 120 minutes for overall convergence criterion of 0.0001. Those are shown in Fig 0.6 and Fig 0.7.

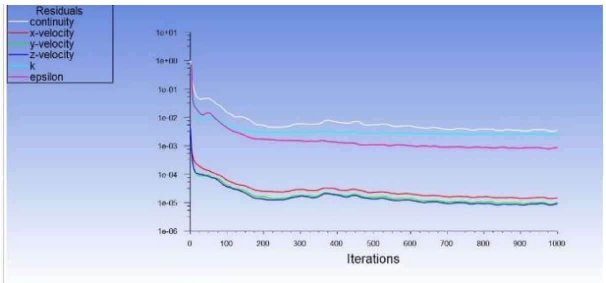


Fig 0.6 1000 iterations residual graph for base model.

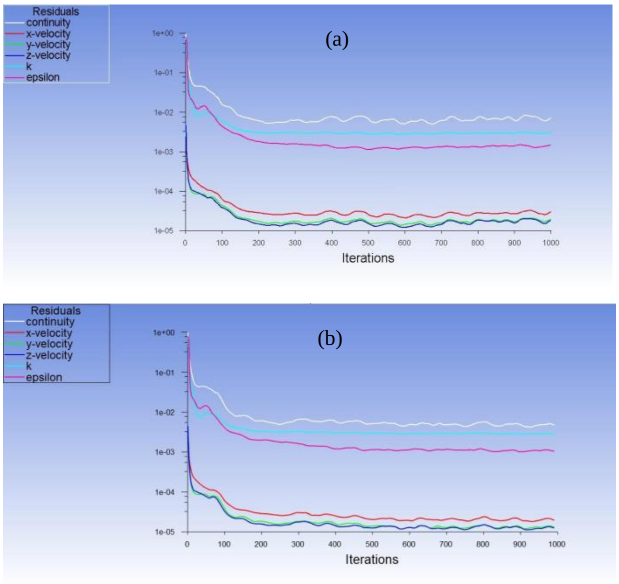


Fig 0.7 Residual graph for (a) one bump & (b) two bumps.

2) ***Residual Characteristics in Stream wise Direction:*** The flow characteristics has found similar to the span wise direction with converged criterion of 0.0001 for 1000 iterations which are shown in Fig 0.8.

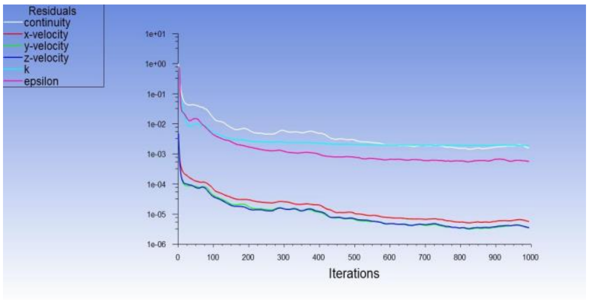
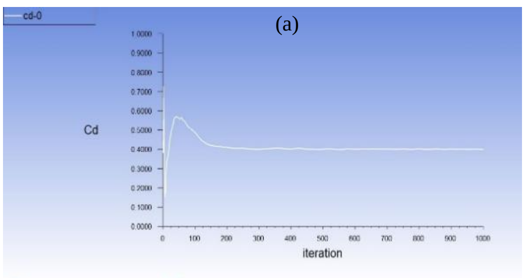


Fig 0.8 Residual Characteristics for Stream wise Two Bumps.

3) ***Drag Co-efficient in Span wise Direction:*** Reduction in the co-efficient of drag was maximum for using two bumps on the rear roof also using one bump has shown an effective result also which is 0.3859 for using one bump and 0.3843 for using two bumps where the base drag co-efficient was 0.4001. These results can be verified by the drag co-efficient graph which are shown in Fig 0.9.



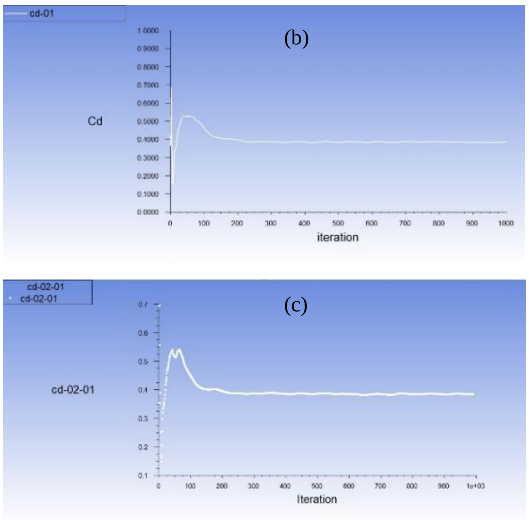


Fig 0.9 Drag co-efficient graph for (a) base model (b) using one bump & (c) two bumps in spanwise direction.

4) ***Drag Co-efficient in Stream wise Direction:*** Further investigation has done for two bumps in stream wise direction, which has shown a satisfactory reduction in drag which is 0.3847 and shown in Fig 0.10.

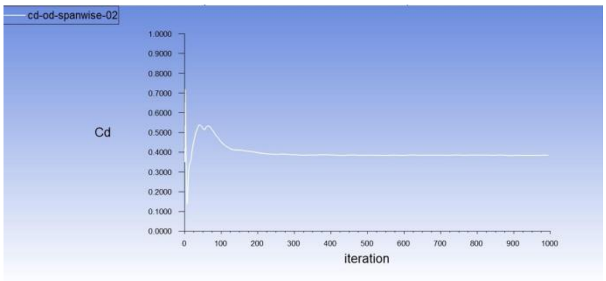


Fig 0.10 Drag co-efficient graph for two bumps in streamwise direction.

V. RESULTS AND DISCUSSION

A. Pressure Distribution and Vector-flow

1) ***Base Model:***

The pressure distribution on the rear roof, front grille, and the wheels are clearly visible and the pressure is maximum at the front grille and low pressure region has been created at the back which is creating pressure gradient difference shown in Fig 0.11 and the flow can be seen to be detached on the roof which is creating strong vortices at the back shown in Fig 0.12. So it indicates an improvement in drag reduction and in fuel economy.

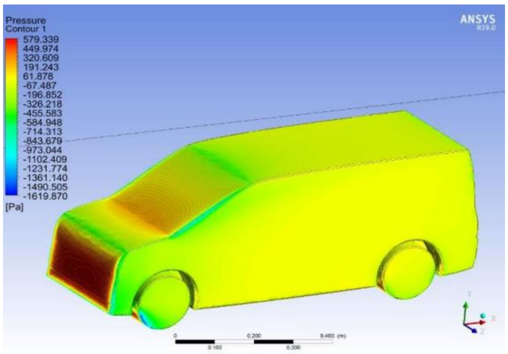


Fig 0.11 Pressure distribution on NOAH 2020 base model.

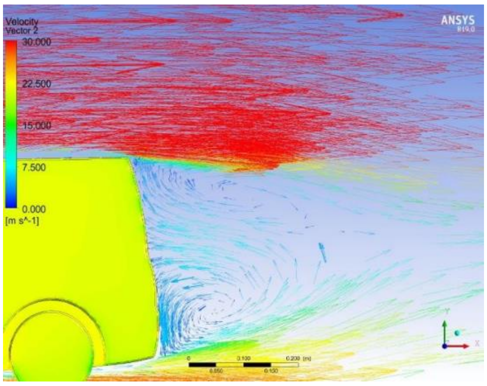


Fig 0.12 Vector flow characteristics on base model.

*2) **Span wise 2 bumps:***

To reduce the drag of base model a simple modification has been done by adding contour bump on the rear roof. After adding bumps the flow over the roof are much tended to attach with the bump and reduces flow separation which led to the reduction in drag. The drag is reduced maximum for using two bumps in span wise direction and the pressure distribution over the bumps and vector flow characteristics are shown in Fig 0.13 and Fig 0.14.

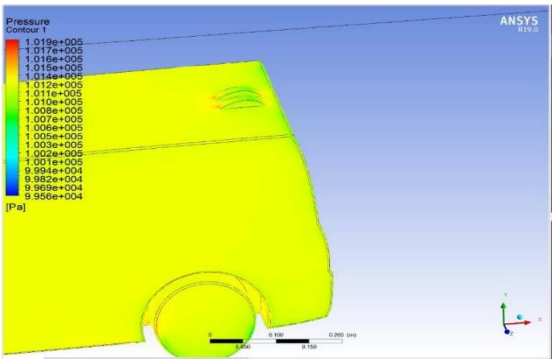


Fig 0.13 Pressure distribution over two bumps in span wise direction.

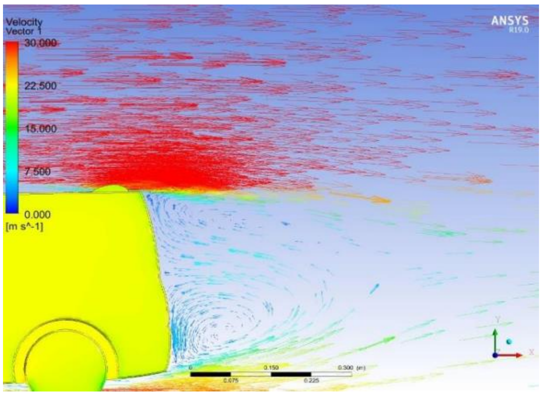


Fig 0.14 Velocity vector flow after adding two bumps.

*3) **Stream wise 2 bumps:***

For further modification, another investigation has been done by placing the two bumps in stream wise direction and it also shows very similar and close characteristics like span wise which are shown in Fig 0.15 and Fig 0.16. It is very clear to see that the flow is much tended in downward direction which also create some downward drag.

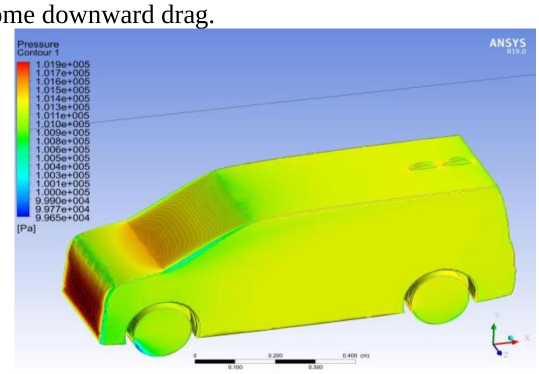


Fig 0.15 Pressure distribution on stream wise directed bumps.

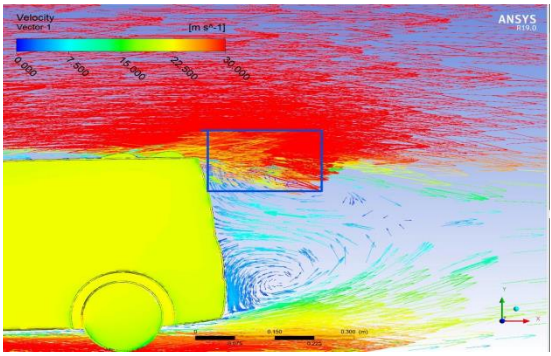


Fig 0.16 Vector flow over stream wise directed bumps.

B. *Drag Calculation*

The overall drag force and the co-efficient of drag is calculated by using the drag formula [9] by considering the frontal area and speed of the vehicle. TABLE VIIVIIIIX shows the overall drag force and drag co-efficient for base model and the suggested models using two bumps.

$F_D = (1/2) \times \rho \times A \times C_d \times V^2$ …………… (1)

$C_d = \frac{FD}{\frac{1}{2}\times\rho\times v^2\times A}$ ………………………… (2)

Where,
F_D = Drag Force
P = Density of air
V = Speed of the vehicle
C_d = Drag co-efficient &
A = Frontal area

TABLE XXIXII
DRAG FORCE AND DRAG CO-EFFICIENT

Experimental Model (s)	**Drag Force (F_D)**	**Drag co-efficient (C_d)**
Base Model	30.52	0.4001
Span wise 2 bumps	30.18	0.3843
Stream wise 2 bumps	30.21	0.3847

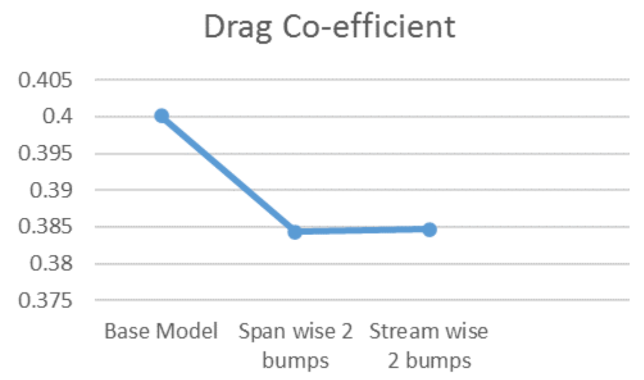


Fig 0.17 Drag co-efficient comparison of suggested models with base model.

It can be seen from the Fig 0.17 that both the span wise and stream wise directed bumps shows very similar and close results in reduction of drag but the maximum reduction occurs in span wise two bumps investigation.

C. Fuel Consumption

Maximum fuel consumption can be reduced by maximum reduction in drag and by considering the fuel factor 0.6 the fuel consumption can be calculated for all the investigations by using fuel consumption formula [10].

$F.C = 3/5 \times \%\Delta C_d$ …….. (3)

TABLE XIIIXIVXVXVI
DRAG FORCE AND DRAG CO-EFFICIENT

Modified Model (s)	**% of drag reduction**	**% of fuel consumption reduction**
Span wise 2 bumps	3.95%	2.37%
Stream wise 2 bumps	3.85%	2.31%

TABLE XVIIXVIIIXIXXX shows the best percentage reduction in drag for the both suggested investigation of using two contour bumps in span wise and stream wise direction and the maximum fuel consumption is reduced for span wise direction which is 2.37% which can be seen from Fig 0.18.

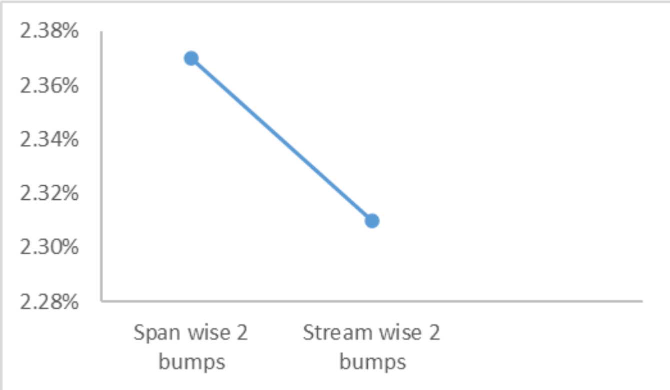


Fig 0.18 Percentage reduction in fuel consumption.

VI. CONCLUSIONS

From the above numerical investigations, we can conclude with the improving aerodynamic efficiency and fuel consumption performance without changing the existing design. The add-on devices like bump has improved the drag coefficient from 0.4001 to 0.3843. Which shows a 3.95% improvement in drag reduction for using 2 bumps on the rear roof of the NOAH 2020 in span wise direction. Whatever the percentage of drag reduction is, it will definitely improve the fuel efficiency and overall power management. Because if the drag is reduced then the car is required less fuel to be burnt for producing maximum power.

Then for further modification we tried stream wise flow by configuring the bumps and found the maximum 3.85% reduction in drag for two bumps. In both cases the drag is tended to increase with the increase in bumps number. Both the investigation has showed an effective reduction in fuel consumption of 2.37% and 2.31% for both span wise and stream wise configuration. This is an effective results for this present situation of fuel economy growth rate all over the world where 1% reduction in fuel consumption can reduce the cost of fuel consumption and helps to national economy. Further modification can be done to improve those results.

ACKNOWLEDGMENT

Special thanks to the whole group members for their hard work during the investigations and others those who have supported by their guidance. This work is supported by the

aerodynamic members of City University, Savar, Dhaka, Bangladesh.

REFERENCES

[1] M. Koike, T. Nagayoshi, and N. Hamamoto, "Research on Aerodynamic Drag Reduction," *Mitsubishi Mot. Tech. Rev.*, vol. No. 16, no. 2, pp. 11–16, 2004.

[2] J. L. Aider, J. F. Beaudoin, and J. E. Wesfreid, "Drag and lift reduction of a 3D bluff-body using active vortex generators," *Exp. Fluids*, vol. 48, no. 5, pp. 771–789, 2010, doi: 10.1007/s00348-009-0770-y.

[3] S. O. Kang *et al.*, "Actively translating a rear diffuser device for the aerodynamic drag reduction of a passenger car," *Int. J. Automot. Technol.*, vol. 13, no. 4, pp. 583–592, Jun. 2012, doi: 10.1007/S12239-012-0056-X.

[4] J. Marklund, L. Lofdahl, H. Danielsson, and G. Olsson, "Performance of an Automotive Under-Body Diffuser Applied to a Sedan and a Wagon Vehicle," *SAE Int. J. Passeng. Cars - Mech. Syst.*, vol. 6, no. 1, pp. 293–307, 2013, doi: 10.4271/2013-01-0952.

[5] A. Ait Moussa, J. Fischer, and R. Yadav, "Aerodynamic Drag Reduction for a Generic Truck Using Geometrically Optimized Rear Cabin Bumps," *J. Eng. (United Kingdom)*, vol. 2015, 2015, doi: 10.1155/2015/789475.

[6] S. M. Mohammed Ali and J. Mohammed Zayan, "Drag Reduction for a Fast Back Passenger Car (Logan)," *Sci. Herit. J.*, vol. 1, no. 2, pp. 19–22, 2017, doi: 10.26480/gws.02.2017.19.22.

[7] A. J. and F. A. Akshoy Ranjan Paul1)*, "DRAG REDUCTION OF A PASSENGER CAR USING FLOW CONTROL TECHNIQUES," *Int. J. …*, vol. 20, no. 2, pp. 397–410, 2019, doi: 10.1007/s12239−019−0039−2.

[8] M. Hariharan, E. Harish Babu, S. Kirubakaran, and S. Gopalakrishnan, "Drag Reduction on Passenger Car," *M.Kumarasamy Coll. Eng.*, vol. 25, no. 6, pp. 4501–4507, 2021, [Online]. Available: https://annalsofrscb.ro/index.php/journal/article/view/6282/4770.

[9] G. Sivaraj, K. M. Parammasivam, and G. Suganya, "Reduction of aerodynamic drag force for reducing fuel consumption in road vehicle using basebleed," *J. Appl. Fluid Mech.*, vol. 11, no. 6, pp. 1489–1495, 2018, doi: 10.29252/jafm.11.06.29115.

[10] and M. A. H. M. H. Pranta*, M. S. Rabbi, "ICMERE2019-PI-181 A COMPUTATIONAL STUDY IN FUEL ECONOMY IMPROVEMENTS FOR HIGHWAY BUS IN BANGLADESH," vol. 2019, pp. 11–13, 2019.