Modification of Small Covered Van to Increase Fuel Efficiency Using Simulation

By

Mahmuda Khatun Mukta

Roll No. 1551506

A thesis submitted in partial fulfillment of the requirements for the degree of

Master of Science in Mathematics

Khulna University of Engineering & Technology

Khulna-9203, Bangladesh

February 2017

Modification of Small Covered Van to Increase Fuel Efficiency Using Simulation

Mahmuda Khatun Mukta

Roll No. 1551506

DEPARTMENT OF MATHEMATICS

KHULNA UNIVERSITY OF ENGINEERING & TECHNOLOGY

Khulna, Bangladesh

Declaration

This is to certify that the thesis work entitled "**Modification of Small Covered Van to Increase Fuel Efficiency Using Simulation"** has been carried out by Mahmuda Khatun Mukta, Roll No.-1551506 in the Department of Mathematics, Khulna University of Engineering & Technology, Khulna, Bangladesh. The above thesis work or any part of this work has not been submitted anywhere for the award of any degree or diploma.

Signature of Supervisor

Signature of Student

Approval

Dedication

To

My beloved parents

S.M. SHAHADAT HOSSAIN

&

LUTFUNNESA

ACKNOLEDGEMENT

First and foremost, praises and thanks to the Almighty Allah for his blessing upon me for the successful completion of this work.

I would like to express my deepest gratitude and appreciation to her honorable supervisor Dr. Mohammad Arif Hossain, Professor, Department of Mathematics. Khulna University of Engineering & Technology (KUET), Khulna, under whose guidance the work has been accomplished. I would also like to thank Prof. Hossain for his earnest feelings and help in matters concerning my research affairs as well as personal affairs.

Dr. Abul Kalam Azad, Principal, Rayermohal (Honours) College, Khulna has done his Ph.D. in the field of drag reduction of moving vehicle. While I was trying to carry on research in the similar field he came forward with all sorts of resources he had. He helped me from choosing a vehicle up to the preparation of the draft thesis. In this context, I want to express my deepest sense of gratitude to Dr. Azad. In fact whatever I have learned in the field of my research I owe to him. It was not possible for me to complete my research without his suggestions, active help and many other roles he has played during the research period.

I thank all other teachers of the Department of Mathematics, KUET for their necessary advice and cordial cooperation during the period of study. I thank all the research students of this department for their help in many respects. My profound debts to my beloved parents are also unlimited. Finally, I would also like to thanks to my uncle and brother for their cordial encouragement and help.

Abstract

In gross, burning a certain amount of fuel in the similar environmental scenario if a vehicle can move more with respect to other vehicle is said to be more fuel efficient. This fuel efficiency certainly has dependency on the drag experienced by the vehicle. So if the drag on the vehicle can be reduced, automatically the fuel efficiency of the vehicle will be increased. There are two way to control the drag, passive and active control. To test the effectiveness of the drag control can be done either through simulation or by direct testing. In this research a passive control is used, inserting some additional shape on the small covered van and is tested through simulation. Three different shapes have been added in the simulation to get idea to reduce drag. For the simulation purpose a CFD software has been used and the result supplied by the software is presented as the result. It has been found that using this passive strategy the drag is reduced. Thus by introducing of some modification the fuel efficiency can be increased.

Contents

List of Tables

List of Figures

CHAPTER 1

Introduction

1. Research Background and Motivation

1.1 History of Vehicle Aerodynamics: A General Background

There are many interesting phenomena appearing in the laminar flow with the growth of Reynolds number. For some value of the Reynolds number (Re) greater than Re_{crit}, the disturbance introduced into flow, instead of being damped, becomes amplified. The flow became unstable and this value of Reynolds number is called critical. Flow stability analysis is usually performed for simple geometries. The nonparallel flow formulation extends the validity of the analysis to general flow. Study of vehicle aerodynamics first began during the earlier part of the 20th century and has continued up until the present day. During the earlier part of the 20th century, vehicle aerodynamics study was associated with vehicle performance, Hucho (1998). Drag reduction, more efficient engine technology and weight reduction, becomes the primary design goals for vehicle engineers and designers around the world in the 1960's to 1980's. Average drag coefficient for typical cars dropped substantially from around 0.5 in the 1960's to typically 0.3 in the late 1980's and mid 1990's Hucho (1998), Wathkins and Alam (2012). At low speeds the main source of drag is the rolling resistance. To achieve high vehicle performance, much of the attention focuses on lowering the vehicle drag coefficient (C_D), which accounted to about 75% to 80% of total motion resistance at 100km/h, Hucho (1998). However, in the last part of the 20th century, during the oil crisis of 1973-1974, the focus on vehicle aerodynamics study shifted towards lowering the drag coefficient in order to produce vehicles with better fuel economy, Hucho (1998). The trend shifted again in the early 1990's especially in North America where a low fuel price coupled with the increased popularity of light trucks and sport utility vehicles have reduced the importance the need on research to reduce drag coefficient, George (1997).

According to the conclusions of International Energy Agency in World Energy Outlook 2007, the gas emissions with greenhouse effect will increase close to 57% in 2030 with strong effects on the environment and the climate.

The human activities became main cause of the increase of the greenhouse gases effect and average global temperature. The activities included the transportation sector where the growth number of automobile is rapidly increasing and make the fuel consumption increases as well. It tends to create harmful effect on the environment because it increases air pollution in the world. Based on these problems it has become a must for automobile industry in the word to immediately create an environmentally friendly automobiles and efficient in fuel consumption.

Fuel consumption of automobile is related to its aerodynamics drag, and the magnitude of aerodynamic drag is highly influenced by separation flows around its shape. Meanwhile, the flow around a traveling automobile is complex and presents nonlinear interactions between different parts of the automobile so that many research institutions and industrial laboratories have been focusing their investigations automotive aerodynamics with numerical studies, Gad-El-Hak (1996). It is necessary to modify locally the flow, to remove or delay the recirculation zone at the back end of the separated swirling structures. This can be mainly obtained by controlling the flow near the wall with or without additional energy using active or passive devices, Fielder and Femholz (1990). Harinaldi et al (2011) worked on the reversed Ahmed body (A vehicle shape accepted by the consortium of 'Model for Vehicle Aerodynamics' to study introduced by S.R. Ahmed and G. Ramm in 1984); they used an active flow control solution by suction and blowing to reduce the aerodynamic drag. The maximum drag reductions associated with these modifications are close to 15.83%.

The optimization of vehicle shapes and the incorporation of commonly used passive control devices have already brought about a significant aerodynamic drag reduction (from C_x= 0.45 in 1975 to C_x = 0.35 in 1985, C_x being the average drag coefficient), Hucho (1998) and Gad-El-Hak (1996). The need to further reduce fuel consumption and provide automobile designers with more creative liberty is prompting the automobile industry to develop innovative active flow control solutions. In the light of such solutions, Glezer and Amitay (2002), used an external energy source to modify the topology without necessarily modifying the shape of the vehicle. Different control techniques have been analyzed in universities and industrial laboratories, and significant results have been obtained on academic geometries, Glezer and Amitay (2002). Continuous suction and or blowing solutions offer a promising alternative, Kourta and Vitale (2008) and seen well- adapted to the automobile context, Gillieron et al. (2002). For example, the efficiency of a suction system in controlling the separation of the boundary layer has been highlighted experimentally on a cylinder by Bourgois and Tensis (2003) and Fournier et al. (2004). The results indicate that significant drag reductions, close to 30%, are obtained by moving the flow separation downstream. Similar results are obtained by Roumeas et al. (2005) on a simplified 2D fastback car geometry.

However, to be practically implemented in controlling the flow separation in the automotive application the passive control methods still need further comprehensive investigations to obtain some fundamental insights of the governing mechanism of separation control. Hence, the current investigation as a part of a long-term fundamental investigation to develop a passive control to the turbulent flow separation which is a fundamental phenomenon governing the aerodynamics performance of vehicle body. The wake flow behind the car is the region which presents the major contribution to the drag and which poses severe problems to numerical predictions and experimental studies as well. The location at which the flow separates determines the size of the separation zone, and consequently the drag force. Clearly, a more exact simulation of the wake flow and of the separation process is essential for the correctness of drag predictions. However, a real life automobile is a very complex shape to model or to study experimentally. Craft et al. (2001) compared the performance of linear and nonlinear $k-\omega$ model with two different wall functions. According Azad et al. (2012) near the body the performance of the $k-\varepsilon$ model is best among the models $k-\varepsilon$, $k-\omega$, Shear Stress Transport (SST) and Baseline $k-\omega$ (BSL).

In this study the covered van used as reference geometry, is modeled to shift the flow around the body for reducing drag. Here different shaped sheets are used from the top of the cab to the top of the storage of the covered van and in front of the storage for controlling the flow around the body with $k-\varepsilon$ turbulence model. As a result change in flow around the covered van is found in some cases that caused the drag to decrease and only in one case drag is increased. Essentially when drag will be reduced without interrupting the mass of the vehicle too much then consumption of fuel will also be reduced.

1.2 Vehicle Aerodynamics: Numerical and Computational Evaluation Methods

Numerical evaluation methods involving vehicle aerodynamics can be done either analytically or by using Computational Fluid Dynamics (CFD). Analytical methods in solving airflow behavior realistically can be done on simple generic type flow problems in either two-dimensional or three-dimensional form. As airflow behavior gets more complex when subjected to flow around complex geometrical domain or bluff bodies, (with the presence of turbulence or compressibility effect), solution of airflow properties cannot be done analytically. This is because in order to obtain its complete turbulent and aerodynamic properties, full unsteady Navier-Stokes (taking into account inertia, viscous and pressure forces) together with the continuity equation (mass conservation) need to be solved. However, obtaining direct numerical solutions of Navier-Stokes equations are still not yet possible even for modern day computers. The main reason being that grid positions needed for a typical CFD model to be solved are Re $^{9/4}$. For a typical flow with Reynolds number of 10^6, it will take the computer to generate and solve equations for $3.16\text{x}10^{13}$ gird points. This is far beyond the reach of even the most state of the art supercomputers available in the world today. In order to come up with a comparable solution, steady or time averaged Navier-Stokes equation is used (called Reynolds Average Navier-Stokes equation – RANS) together with turbulence model, developed to take into closure problems involving Reynolds stresses resulting from the time averaging process.

CFD approach for turbulence modeling was first intended for the aerospace community in the 1960s and 1970s (Anderson, 1995). In the early development stage of CFD for automotive applications, codes were expected to provide actual quantitative data that is similar to measured wind tunnel data. Knowing that this is not yet possible, present use of CFD in automotive are used to provide information about flow characteristics and phenomena, which dictates aerodynamic performance. However, the ultimate goal in CFD is to obtain model of flow as actual as possible and current and future research on CFD is ongoing in order to achieve that goal. Furthermore, current applications of CFD in the automotive industry are determined by economic viability. To be economically viable, the codes should be able to simulate the correct physics of the flow and at the same time achieve computational turnaround time that is the same or less than that of a wind tunnel Test cycle time. Ahmed (1998) has showed that for a typical vehicle, current testing time taken in a wind tunnel in order to achieve desired level of C_D reduction has increased. This will be an expensive exercise for automobile manufactures. With the reduction on computational cost, aerodynamic simulation by using CFD, being run at a faster turnaround time will only be at a fraction of the cost.

However, this will only put more demand on the current performance of computers speed and memory. These are due to several factors.

- An increase in sophistication of flow physics modeled.
- An increase in modeled geometries complexities.
- An increasing number of multidisciplinary approaches of flow simulation.

These increases in computational demands are intended in achieving the ultimate goal- fluid flow realism, in CFD simulation as mentioned earlier. Much more complex three-dimensional vehicle geometries are now being used in automotive CFD simulation coupled with high grid density to achieve better flow resolution. Usually this also leads to a more accurate and realistic flow simulation. In an unsteady three-dimensional flow, a doubling of grid density (to double the accuracy) results in a (with three space coordinates and a time dimension) sixteen-fold increase in computation effort. In addition, better CFD post processing flow visualization effect such as colour-coded pressure distributions over the entire body surface and observation of particle traces in real time animation also puts extra demand on computer speed and memory.

1.3 Turbulence

Turbulence in a fluid refers to three dimensional, unsteady motions of particles that are practically in chaotic manner. It appears in the flow field as a random process that is completely unpredictable. This is quite in contrast with the laminar state, where flow is apparently uniform and well-behaved. In a large variety of applications, it is possible to identify a base flow that is reasonably well-behaved and superimposed on which are random fluctuations in flow properties.

Turbulences have their great importance in computational fluid dynamics (CFD). The understanding of the physics of turbulence is crucial and many different models have evolved to explain them. Sometimes the turbulence models are validated through vehicle aerodynamics. For many years computers have been used to solve fluid flow problems. Numerous programs have been written to solve either specific problems, or specific classes of problems. In the mid-1970’s general purpose CFD solvers were started to develop. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labor intensive and reducing time. Robust solutions of the flow field can be achieved through advanced solvers that contain algorithms which enable to solve the problem in a reasonable time.

1.4 Some Important Types of Flows

(a) Laminar and turbulent flows: A flow, in which each fluid particle traces out a definite curve and the curves traced out by any two different fluid particles do not intersect, is said to be laminar. On the other hand, a flow, in which each fluid particles does not trace out a definite curve and curves traced out by fluid particles intersect, is said to be turbulent. Practically in turbulent flow the fluid velocity varies very rapidly in an irregular manner. In other words, turbulence in a fluid refers to three dimensional, unsteady motions of particles that are practically in chaotic manner. Turbulent fluid motion is an irregular condition of flow in which the various quantities show a random variation with space and time coordinates, so that statistically, only distinct average values can be discerned (Hinze, 1959). The origin of turbulence lies in a region in the physical or parameter space adjacent to the laminar regime and is called as the transition region.

(b) Steady and unsteady flows: A flow in which properties and conditions associated with the motion of the fluid are independent of the time so that the flow patterns remain unchanged with respect to time, is said to be steady. On the other hand, a flow, in which properties and conditions associated with the motion of the fluid depend on the time so that the flow pattern varies with time, is said to be unsteady.

(c) Homogeneous isotropic turbulence: The peculiarity of turbulence is that its behavior in different scales- both of time and of space is different, and it has been amply demonstrated that the smaller scales demonstrate a considerable degree of isotropy, for they are less affected by the boundaries of the flow. Homogeneity, again though rare in the large scale, is often present in the smaller scales. So the study of homogeneous isotropic flows is not altogether wasted even if we consider real flows. Homogeneity is really a contraction of ‘spatial homogeneity’ and indicates that mean properties do not vary with absolute position in a particular direction.

1.5 Physics of Turbulent Motion

Turbulence in fluids must satisfy the laws of classical physics, namely conservation of mass, Newton’s second law of motion and conservation of energy. The fact that turbulence is chaotic shows that these constraints are not sufficiently strong to generate a unique flow field. Recently it has been originated to analyze turbulence as the result of a sequence of bifurcations of a low order dynamical system that is governed by a system of first order non-linear ordinary differential equations. These research initiatives throw light on details of physical processes observed in turbulent flow fields. An understanding in terms of turbulence physics is quite useful in evaluating models or developing new ones.

1.6 Numerical Computation of Turbulent Flows

The problem of calculating turbulence can be seen as part of the more general field of computational fluid dynamics (CFD). CFD is extensively concerned with the numerical representation and computation of the partial differential equations which govern the motion of real fluids. The subject tends to divide into two areas: the development of numerical methods, and the creation of algorithms to implement these methods. At the same time, progress in CFD must necessarily depend on developments in computing process. In particular, we should perhaps mention the growing use of concurrent computer architectures, which offer large increases in memory and speed. In the present day various type of turbulence models develop for simulating turbulent flows.

1.7 Vortex or Rotational Motion

Rotational motions differ from potential flows in that, as the name applies, all particles of the fluid or at least part of them rotate about an axis which moves with the fluid. Potential flow on the other hand, is irrotational by definition.

1.8 Body and Surface Force

In the study a fluid dynamics we distinguish between two types of forces acting on a fluid element, namely, body forces and surface force. The body forces are distributed throughout the volume of the body, and there are usually expressed as 'force per unit mass of the element’. Examples are gravity and inertia forces. Any force acting over the surface area is called the surface force. Surface force arises due to the action of surrounding fluid on the element under consideration through direct contact.

1.9 No slip Condition

This is the most common type of wall boundary condition. A real fluid, the existence of intermolecular attractions causes the fluid to adhere to a solid wall and this gives rise to shearing stress. The inner layers of a real fluid transmit tangential and normal stresses. On the boundary between a perfect fluid and a solid wall there exists a difference in relative tangential velocities i.e. there is slip. The existence of tangential (shearing) stresses and the condition of no slip near solid walls constitute the essential differences between a perfect and a real fluid. The real fluids have a zero velocity at the walls and hence they cannot slip at the boundary wall. This is known as no slip condition.

1.10 Boundary Layer Theory

(a) Prandtl's boundary layer theory: For convenience, consider laminar two-dimensional flow of fluid of small viscosity (large Reynold's number) over a fixed semi-infinite plate. It is observed that, unlike an ideal (non-viscous) fluid flow, the fluid does not slide over the plate, but “sticks” to it. Since the plate is at rest, the fluid in contact with it will also be at rest. As we move outwards along the normal, the velocity of the fluid will gradually increase and at a distance far from the plate the full stream velocity is attained. However, it will be assumed that the transition from zero velocity at the plate to the full magnitude takes place within a thin layer of fluid in contact with the plate. This layer is known as the boundary layer.

(b) Importance of Prandtl’s boundary layer theory in the fluid dynamics: Although the boundary layer is thin, it plays a vital role in fluid dynamics. It has become a very powerful method of analyzing the complex behavior of real fluids. The concept of a boundary layer can be utilized to simplify the Navier-Stokes equations to such an extent that it becomes possible to take many practical problems of great importance. The drag on ships and vehicle, the efficiency of compressors and turbines in jet engines, the effectiveness of air intakes for ram and turbojets and so on depend on the concept of the boundary layer and its effects on the main flow. The boundary layer theory is able to predict flow separation. It can explain the existence of a wake.

(c) Boundary layer thickness: The boundary layer thickness is defined as the elevation above the boundary which covers a region of flow where there is a large velocity gradient and consequently non-negligible viscous effect.

The boundary-layer thickness, $\delta_{ij} = \sqrt{\left(\frac{\mu x}{\rho U_{\propto}}\right)}$,

where $U_{\propto}$ is the velocity of the outer flow, ρ is the density, μ is coefficient of viscosity and x is the length of the plate.

1.11 Some Important Non dimensional Quantities

(a) Reynold's number, Re: It is the most important parameter of the fluid dynamics of a viscous fluid. It represents the ratio of the inertia force to viscous force a and is defined as

Re = inertia force /viscous force = $\frac{\rho U^2 L^2}{\mu U L} = \frac{UL}{\upsilon}$

where U, L, ρ, μ and *v* are the characteristic value of velocity, length, density, coefficient of viscosity or coefficient of dynamic viscosity and coefficient of kinematic viscosity of the fluid respectively. When the Reynolds number of the system is small the viscous force is predominant and the effect of viscosity is important in the whole velocity field. When the Reynolds number is large the inertia force is predominant, and the effect of viscosity is important only in a narrow region near the solid wall or other restricted region which is known as boundary layer. If the Reynolds number is enormously large (Re≥2000), the flow becomes turbulent. A critical Reynolds number Re_{crit} = $5X10^5$ is visible in the drag coefficient C_D (Re) of a flat plate at zero incidence. For Reynolds numbers which are smaller than Re_{crit}, the flow past the plate is laminar: and above Re_{crit} the flow past the plate is turbulent.

(b) Prandtl number, Pr: The Prandtl number is defined by

$Pr = \mu g c_p/k$,

which is the ratio of kinematic viscosity to the thermal diffusivity,

where μ is the coefficient of viscosity, g is the acceleration due to gravity, c_p is the specific heat at constant pressure and k is the thermal conductivity. Evidently Pr depends only on the properties of the fluid. For air Pr = 0.7 approx. and for water (at 60°F) Pr = 7 approximately.

(c) Mach number, M: The mach number M is defined by M $= {}^{q}/_{a}$, where q is the velocity of flow and *a* is the velocity of sound. Mach number is also expressed in terms of the ratio of inertia force and the elastic force. When the Mach number is small (i.e., M<<1), the fluid can be taken as incompressible. On the other hand, if Mach number is nearly one or greater than one, the fluid will be taken as compressible.

(d) Euler number Eu: Eu = pressure force / Inertia force = $P/V^2\rho$

where P, V and ρ are the characteristic pressure, characteristic velocity and density respectively. When the pressure force is the predominating force, Euler's number must be the same for dynamic similarity of two flows.

1.12 Grid Points

To compute a turbulent flow by directly using the Navier- Stokes equations would require us to use approximately $Re^{3/4}$ grid points in each direction and about $Re^{1/2}$ time-steps. So for a typical Reynolds number of 10^6 we would need more than 10000 grid points in each direction (or a total of 10^{12} grid points for a three dimensional calculation) and more than 1000 time-steps to get a reasonable simulation of the flow.

1.13 Turbulence Modeling

Generating turbulence information by solving the full Navier-Stokes or Reynolds-Stress equations remains incomplete at the time. Instead, analysts resort to approximate approaches, called as modeling. Turbulence modeling is based on the assumption that the real flow field may be substituted by an imaginary field of mathematically defined continuous functions. These functions usually represent physical quantities measurable in the flow field. Many turbulence modeling techniques deal with approximation to the Navier-Stokes or Reynolds-Stress equations. Any model, up to some extent can be analytically derived from Reynolds-stress equations. The main goals of turbulence modeling are: to develop a set of constitutive relations valid for any general turbulent flow problem; yield sufficiently reliable answers and offer a degree of universality sufficient to justify their usage in comparison to cheaper, less general methods or to more expensive but potentially more reliable methods.

Early work on modeling turbulence was attracted by Newton's law of viscosity. Eddy viscosity a new property of turbulence was introduced and specified for different turbulent flows. Many simple models based on the eddy viscosity concept, particularly Prandtl mixing length models were developed to predict the mean velocity profiles in turbulent flows. These models continue to be in use because of their simplicity and sufficient accuracy in determining global quantities such as boundary-layer thickness, wall shear stress and point of separation. Use of these models produced analytical solutions for many simple engineering problems. Advanced engineering applications require identification of structures and calculation of statistical parameters, spectral functions, Reynolds stress distribution and turbulence heat and mass flux distributions. The model that must then be selected depends on the level of detail to be captured by the solution.

The k-ε Model:

Turbulence model has two tasks-

(a) to relate the Reynolds stress term $\overline{u_i' u_j'}$ to the turbulent parameters and to the mean flow field

(b) to determine the distribution of the parameters over the field.

Many models employ the eddy-viscosity concept, which is given by

$$-\overline{u_i' u_j'} = v_t \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \quad (1.1)$$

The turbulent viscosity in Equation (1.1), v_t is computed from a velocity scale ($k^{1/2}$) and a length scale ($k^{3/2}$ /ε) which are predicted at each point in the flow via solution of the following transport equations for turbulent kinetic energy (*k*) and its dissipation rate (ε):

$$\frac{\partial k}{\partial t} + \overline{u_i} \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{v_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + v_t \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) \frac{\partial \overline{u_i}}{\partial x_j} - \varepsilon \quad (1.2)$$

$$\frac{\partial \varepsilon}{\partial t} + \overline{u_i} \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{v_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} G - C_{2\varepsilon} \frac{\varepsilon^2}{k} \quad (1.3)$$

where G is the generation of k and is given by

$$G = v_t \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) \frac{\partial \overline{u_i}}{\partial x_j} \quad (1.4)$$

The turbulent viscosity is then related to k and ε by the expression

$$v_t = C_\mu \frac{k^2}{\varepsilon} \quad (1.5)$$

The coefficients C_μ, $C_{1\varepsilon}$, $C_{2\varepsilon}$, σ_k and σ_ε are constants which have the following empirically derived values

C_μ =0.09, $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, σ_k =1.0, σ_ε =1.3

1.14 Drag Coefficient

The drag experienced by a plate is purely friction drag. This can easily be determined from the equation given below.

$$D = b \int_0^l \tau_w(x) dx \quad (1.6)$$

From the equation (1.6) we get the drag of one side of the plate, where b is the width of the plate and *l* its length. Now the local wall shear stress is

$$\tau_w(x) = \mu \left(\frac{\partial u}{\partial y} \right)_w = \mu U_\propto \sqrt{\frac{U_\propto}{2vx}} f''_w = 0.332 \mu U_\propto \sqrt{\frac{U_\propto}{vx}} \quad (1.7)$$

where $f_w^{''}$ is the characteristic value for the boundary layer on a flat plate at zero incidence. The skin-friction coefficient in the equation

$$C_f(x^*) = \frac{2\tau_w(x^*)}{pV^2} \tag{1.8}$$

With the reference velocity $U_\propto$ it becomes

$$C_f(x) = \frac{2\tau_w(x)}{\rho U^2} = \frac{0.664}{\sqrt{Re_x}} \tag{1.9}$$

where the Reynolds number formed with the length x has been used:

$$Re_x = \frac{U_\propto x}{v} \tag{1.10}$$

If we want to estimate the value of drag coefficient in the usual manner, then we use the following equation.

$$C_D = \frac{2D}{pU_\propto^2 bl} \tag{1.11}$$

where the wetted area *bl* serves as a reference area

1.15 Roughness Height

The roughness height is the height of the surface irregularities for uniform sand grain roughness, or a mean height value for non-uniform sand-grain roughness. All surfaces in technical applications like the surface of a car are rough with a deviation in roughness height. We have it even for very smooth surface. In most CFD program a standard roughness is set. But if we have a much rough surface like sand corn size we have to modify the wall functions for the turbulence model with the right roughness height.

CHAPTER 2

Methodology and Governing Equation

2.1 CFD Software

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow (in a special form) over a region of interest with specified conditions on the boundary of region.

The history of CFD

Computers have been used to solve fluid flow problems for many years. Numerous programs have been written to solve either specific problems, or specific classes of problems. From the mid-1970's, the complex mathematics required to generalize the algorithms began to be understood, and general purpose CFD solves were developed. These began to appear in the early 1980's and required what were then very powerful computers as well as in-depth knowledge of fluid dynamics, and large amounts of time to set up simulations. Consequently CFD was a tool used almost exclusively in research.

Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labor intensive, reducing time and hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time.

As a result of these factors Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost effective and accurate alternative to scale model testing, with variations on the simulation being performed quickly, offering obvious advantages.

The set of equations which describe the processes of momentum, heat and mass transfer are known as the Navier-Stokes equations. These partial differential equations were derived in the early nineteenth century and have no known general analytical solution but can be discretized and solved numerically.

Equations describing other processes can also be solved in conjunction with the Navier-Stokes equations. Often, an approximating model is used to derived these additional equations, turbulence models being a particularly important complex.

There are a number of different solution methods which are used in CFD code. The most common and the one on which ANSYS CFX is based, is known as the finite volume technique. In this technique, the region of interest is divided into small control volume. As a result an approximation of the value of each variable at specific points throughout the domain can be obtained. In this way one derives a full picture of the behavior of the flow.

In this simulations Ansys 11© with high resolution advection scheme along with Physical timescale (physical time = length of the tunnel/velocity of the fluid) is used.

2.2 Simulation Setup

2.2.1 Geometry: The geometry of body is shown in Fig. 1. Storage height H=1.3716m, the storage width=1.0741×H and length =1.5185×H. The cab height =0.5926×H, cab width=1.0741×H and the cab length=0.8704×H. The body is placed in the channel with a cross section of B×F=2.1872H× 3.6454H (width × height). The length of the channel =13.1234×H. The body is lifted from the floor, producing a ground clearance of 0.4630×H. **As a result the channel blockage is about 13.4667 %.**

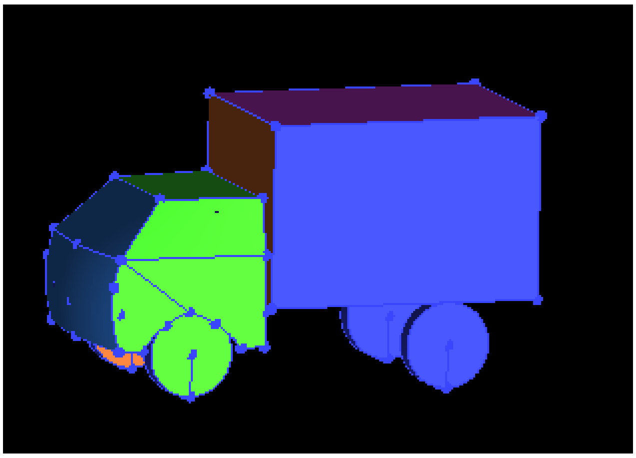

Fig. 1: Geometric presentation of the body.

2.3 Transport Equations

The continuity equation is written as a combination of the transient and convection terms:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_j)}{\partial x_j} = 0 \tag{2.1}$$

The momentum equation is written as a combination of the transient, convection, diffusion and source term:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial(u_j u_i)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial u_i}{\partial x_j}\right) + s_i \tag{2.2}$$

The momentum equation is transformed to the Navier-Stokes equation (for incompressible fluid) and is written in simplified conserved form as:

$$\frac{\partial \mathrm{u_i}}{\partial \mathrm{t}} + u_j \frac{\partial(\rho u_i)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{1}{\rho}\frac{\partial}{\partial x_j}\left(\mu \frac{\partial u_i}{\partial x_j}\right) + \mathrm{s_i} \tag{2.3}$$

The transport equation for the turbulent kinetic energy is:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij}\frac{\partial U_i}{\partial x_j} - v\overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u_i'}{\partial x_k}} + \frac{\partial}{\partial \mathrm{x_j}}\left[\mathrm{v}\frac{\partial \mathrm{k}}{\partial \mathrm{x_j}} - \frac{1}{2}\overline{u_i' u_i' u_j'} - \frac{1}{\rho}\overline{\mathrm{P'u_j'}}\right] \tag{2.4}$$

2.4 Methodology:

To modify the flow a sheet will be placed over the cab up to the storage. Three different position/shape placement of the sheet are studied.

a) A sheet is placed covering half of the cab up to the storage of covered van.
b) A sheet is placed create a add-ons at the top front of the storage whose lower surface is parallel to the horizontal.
c) A sheet is placed create a add-ons at the top front of the storage whose upper surface is parallel to the horizontal.

For the simulation we have scaled the body at 1:10 ratio. Front clearance is 0.4582 times and body to outlet clearance 3.7912 times of the whole body. Channel height is 3.6443 times of the Van height.

Before final simulations grid independence test is done. Turbulence model and roughness height are selected through comparison among different selections.

2.5 Predefined Parameters:

2.5.1 Boundary Conditions

Boundary conditions are a set of properties or conditions on surfaces of domains and are required to fully define the flow simulation. The type of boundary condition that can be set depends upon the boundary surface. A fluid boundary is an external surface of the fluid domain excluding surfaces where it meets other domains.

A fluid boundary supports following boundary conditions:

(a) Inlet-Fluid predominantly flows into the domain.

When we define an area to be an inlet then we anticipate that flow will be in to a domain. We can set either velocity or pressure at the point, and solver will calculate the other value.

In the inlet we applied the Velocity = 40 m/s and temperature = 298K.

(b) Outlet-Fluid predominantly flows out of the domain.

When we define an area to be an outlet then we anticipate that flow will be out from a domain.

The outlet boundary condition applied is: Relative Static Pressure = 0 Pa.

(c) Wall-Impenetrable boundary to fluid flow.

The term impenetrable will mean that through the wall no suction or blowing effect will take place i.e. there will be porosity.

There are three options for the influence of a wall boundary on the flow, namely:

(i) No Slip Wall (ii) Free Slip Wall and (iii) Rotating Wall.

No Slip: This the most common type of wall boundary condition. In a real fluid, the existence of intermolecular attractions causes the fluid to adhere to a solid wall and this gives rise to shearing stress. The real fluids have a zero velocity at the walls and hence they cannot slip at the boundary wall. This is known as no slip condition.

Free Slip: In this case the velocity component parallel to the wall has a finite value (which is computed) but the velocity normal to the wall, and the wall shear stress, are both set to zero.

Rotating Wall: This option applies to both stationary and rotating domains and allows the wall to rotate with a specified angular velocity. The angular velocity is always in relation to the local (relative) frame of reference. An axis must be specified in a stationary domain and can optionally be specified in a rotating domain.

We applied the following wall boundary conditions.

Wall influence on flow = No slip
Wall function = Scalable
Wall roughness = Rough wall
Roughness height = 0.0002m
Fixed temperature = 298K

2.5.2 Simulation Parameters

The following conditions are used as simulation parameters:

Simulation type	Stationary
Domain type	Fluid domain
Flow region	Subsonic
Fluid	Air at 298K
Buoyancy	Buoyant
Reference pressure	1 [atm]
Gravity	X = 0
	Y = 0
	Z = -9.807m/s^2
Buoyancy reference temperature	298K
Domain motion	Stationary
Heat transfer option	Isothermal
Buoyancy turbulence	Production and dissipation,
Turbulence balance	k – Epsilon
Equation class setting	Continuity,
	Turbulence Eddy Dissipation,
	Turbulence Kinetic Energy and
	Momentum
Solver control	
Advection scheme	High resolution
Maximum number of iterations	100

Timescale control	Physical timescale
Physical time	Length of tunnel/velocity of fluid
Residual type	Maximum
Residual target	0.0000001
Conservation target	0.01
Wall function	Scalable
Turbulence	Low(intensity=1%)
Static pressure	0 Pa
Wall influence on flow	No slip
Wall roughness	Rough wall
Roughness height	0.0002m
Fixed temperature	298K

CHAPTER 3

Results and Discussions

3.1 Validation of uncontrolled flow:

Trial and error process has been used to find the number of elements suitable for the generation of covered van. For the purpose extension of near wake separation bubble has been cheeked and it has been found that about 1838596 elements can be handled by the computing facility used and is suitable and comparable to the bench mark, for which we have accepted the work of Alam and Watkins (2013).

The turbulent model has been chosen through experimentation and comparison among the different models. It has been found that k-**ε** model is better suited for numerical simulation over Ahmed car body, Azad et al. (2012).

The roughness height has been taken through experimentatiom and comparison between the k-**ε** model and RNG k-**ε** model. Azad et al. (2013), has shown that while using the k-**ε** turbulence model selection of roughness height of 0.0002m is the better choice.

3.2 Experimental Results

As a passive strategy to control the flow experiments are done using a sheet placed over the cab up to the storage, having different size. It may be noted that for the experimental purpose a gap of 2 inch between the cab and the storage is used. The choice is not accidental, rather the covered van that are running in the roads have this gap introduced by the makers.

Three different size of sheets are used in these experiments –

1) A sheet placed to cover the 1/2 of the cab up to the storage.
2) A sheet is placed to create a add-ons at the top front of the storage whose lower surface is parallel to the horizontal.
3) A sheet is placed to create a add-ons at the top front of the storage whose upper surface is parallel to the horizontal.

For the comparison purpose these simulation along with the uncontrolled flow are labeled as case-1, case -2, case-3 and case-0 respectively. The geometric representations of case-1, case-2, and case-3 are presented in the following figures.

3.2.1. Case-1: A sheet placed to covered the 1/2 of the cab up to the storage.

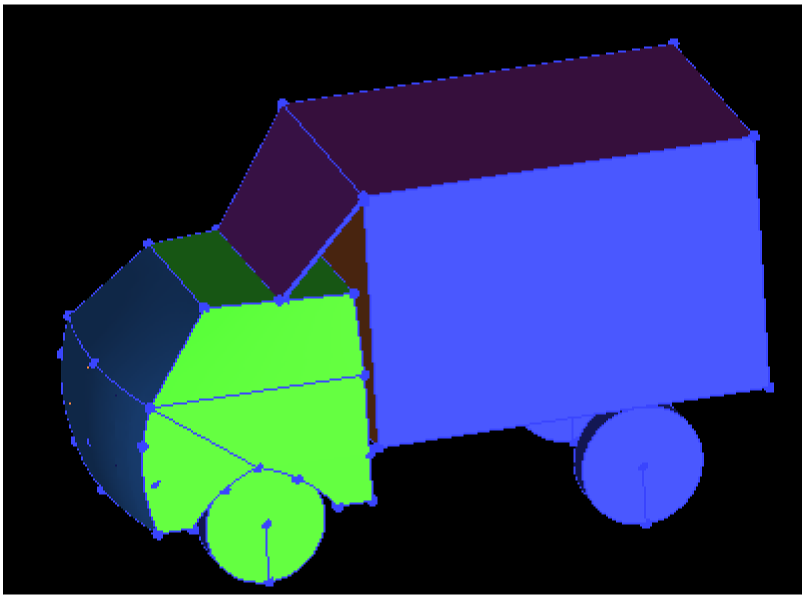

Fig.2: Geometric representation of case-1

3.2.2. Case-2: A sheet is placed to create add-ons at the top front of the storage whose lower surface is parallel to the horizontal.

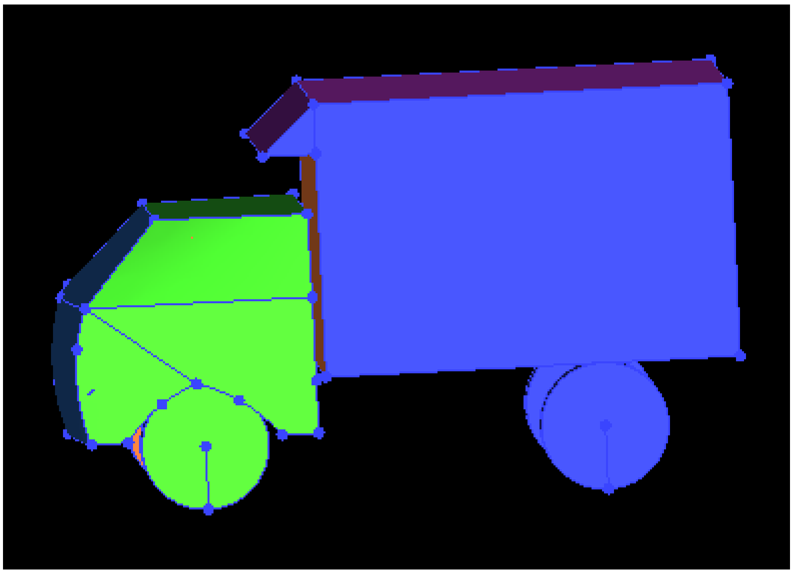

Fig.3: Geometric representation of case-2

3.2.3. Case-3: A sheet is placed create add-ons at the top front of the storage whose upper surface is parallel to the horizontal (case-3).

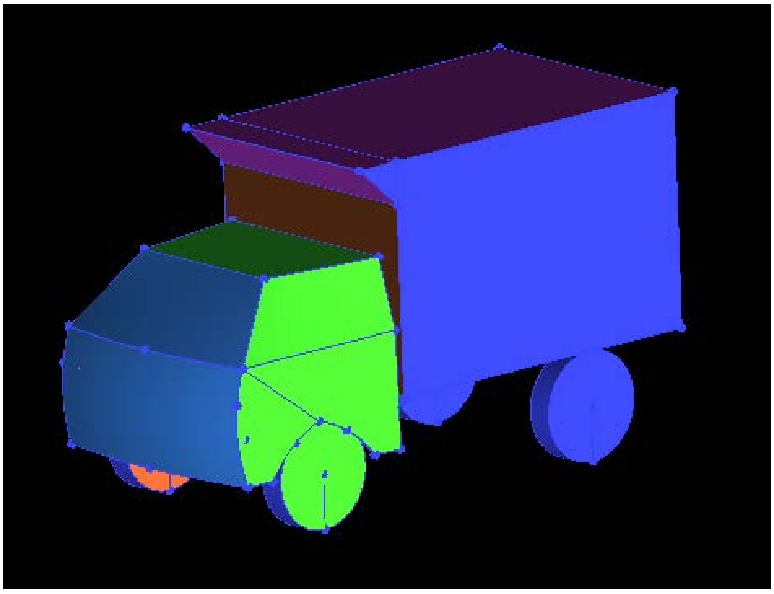

Fig.4: Geometric representation of case-3

Before experimentation the drag coefficients of the uncontrolled flow labeled as case-0, is compared with those of the benchmark results obtained by Alam and Watkins (2013). This comparison presented in Table 3.1 validates the observed drag through simulation.

It has been found that the total drags are comparable as presented in Table 3.1. From the tabulated drag it is observed that the error is about 7.644%

Table 3.1: Validation of drag force (for uncontrolled flow).

	Total drag coefficient	Error %
Alam and Watkins (2013)	0.90000	
k-ε	0.8312	7.6444

Table 3.2: Comparison among the different cases.

Case No	Total drag coefficient	Change %
0	0.8312	0
1	0.4562	45.1155
2	0.7794	6.2320
3	0.8641	-3.9581

Table 3.2 represents the total drag coefficients in different cases. It is seen from Table 3.2 that for case 1 when a sheet is placed to covered the 1/2 of the cab up to the storage then the total drag coefficient decreased maximum. But for the other cases the total drag coefficient increased than case-1. Finally by placing a sheet the total drag coefficient can reduced 45.1155%.

It has been found that about 45.1155% of the total drag can be reduced by placing a sheet, which in turn will reduce a fuel consumption of about 37.5%.

CHAPTER 4

Conclusions

The title of the research expresses the target of the research i.e. finally the drag should be reduced. The title itself express that Covered van is to be used for CFD modeling to reduce drag. Thus Covered van is to be built through introduction of proper number of elements so that its behaviors are comparable with benchmark. Suitable turbulence model is to be chosen so that the uncontrolled flow is to be comparable to the benchmark. The related factors with the turbulence model also may require to be selected. Modification of Covered van is to be done to reduce drag. Passive control i.e. the control which cannot be changed within the whole span of simulation is chosen. Sheets at different locations are used as the control parameter. With the proper selection of the above mentioned items drag reduction is found.

Before starting simulation to reduce drag grid independence, comparison between different turbulence models and selection of roughness height are done. On the basis of the experiments the following conclusions are made, which guided the after wards simulations.

(1) The extension of near wake separation bubble do not changes when number of elements is taken about 1838596 elements.
(2) Out of the $k-\varepsilon$, $k-\omega$, SST and BSL turbulence model k-ε model is best in performance near the body, thus is chosen to calculate the drag.
(3) $k-\varepsilon$ model has dependence on the selection of the roughness height. A roughness height 0.0002m is better suited for drag evaluation over covered van and that is taken.

The simulations by the modified covered van produced different drags, lower than the uncontrolled flow. On the basis of the simulations with the different modifications by inserting sheets of different size and shape the following conclusions can be made:

(1) A maximum of total drag is reduced through placing a sheet on the cab to the storage of covered van.
(2) A maximum of about 45.11% drag reduction is achieved when the sheet has been placed on the middle of the cab to storage of covered van.
(3) As drag has been reduced so fuel consumption to run same distance will also be reduced. Thus in turns fuel efficiency is increased.

Thus finally we may conclude that by placing a sheet on the cab to the storage one can reduce the total drag over covered van and ultimately the fuel consumption.

Reference

Ahmed, S.R. and Ramm, G., “Some Salient Features of Time Averaged Ground Vehicle Wake,” SAE Technical Paper, 840300, 1984.

Ahmed, S.R., “Computational Fluid Dynamics”, Chapter XV in Hucho, W. H. (Ed.), Aerodynamics of Road Vehicles, 4th Edition, SAE International, Warrendale, PA, USA, 1998.

Alam, F. and Watkins, S. “Implication of Vehicle Aerodynamics of Fuel Saving and the Environment”, International Conference on Mechanical, Industrial and Materials Engineering, RUET, Rajshahi, Bangladesh, 2013.

Anderson, J.D. J.r., “Computational Fluid Dynamics: The Basics with Applications”, McGraw Hill, 1995.

Azad, A. K. Hossain M. A. and Islam, A.K.M.S. 2012. “Comparison of Turbulence Models for Ahmed Car Body Simulation”, 5th BSME International Conference on Thermal Engineering, IUT, Dhaka, Bangladesh.

Azad, A. K. Hossain M. A. and Islam, A.K.M.S. 2013. “Effect of Roughness Height on the Turbulence Models for Ahmed Car Body Simulation”, International Conference on Mechanical , Industrial and Materials Engineering, RUET, Rajshahi, Bangladesh.

Bourgois, S. and Tensi, J., “Controle de I’ecoulement par autour d’un cylinder par techniques fluidiques et acoustiques”, 16eme Congres Francais de Mecanique, Nice, 2003.

Craft, T. J. Gant, S.E. Iacovides, H. and Launder,B. E.2001. “Computational study of flow around the Ahmed car body”, 9th ERCOFTAC workshop on refined turbulence modeling, Darmstadt University of Technology, Germany.

Fieldler, H.E. and Femholz, H.E., “On the management and control of turbulent shear flows”, Prog. Aero. Sci, vol. 27,1990.

Fournier, G., Bourghois, S., Pellerin, S., Ta Phuoc, L., Tensi, J. and El Jabi,R., “Wall suction influence on the flow around a cylinder in laminar wake configuration by eddy simulation and experimental approaches”, 39e Colloque d’Aerodynamique Applique, Centrole des ecoulements, Mars, Paris, 22-24, 2004.

Gad-El-Hak, M., “Modern developments in flow control”. Appl Mech Rev.Vol. no.9, pp 365-379, 1996.

George, A.R., (Ed), “Automotive Wind Noise and its Measurement”, An Information Report of the SAEV Wind Noise Measurement Committee, 1997.

Gillieron,P., “Conttrole des ecoulements appliques a I’automobile. etat de I’art”, Mecanique and Industries, vol: 3, pp. 515-524, 2002.

Glezer, A, and Amitay, M, “Synthetic jets”, Annual Review of Fluid Mechanics, vol.34, pp 503-529, 2002.

Harinaldi, Budiarso, Rustan Tarakka and Sabar P. Simanungkalit, “Computational Analysis of Active Flow Control to Reduce Aerodynamics Drag on a Van Model”, International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS Vol: 11 No: 03, pp. 24-30, 2011.

Hinze, J. O. “Turbulence”, McGraw Hill, New York, USA 1959.

Hucho, W.H. (Ed.), “Aerodynamics of Road Vehicles”, 4^{th} Edition, SAE International, Warren dale, PA, USA, 1998.

Kourta, A.and Vitale, E., “Analysis and control of cavity flow”, Physics of Fluids vol. 20: 077104, 2008.

Roumeas, M., Gilleron, P. and Kourta, A., “Reduction de trainee per controle des decollements autour d’une geometric simplifiee: etude parametrique 2D”, 17eme Congres Francais de Mecanique, 29 Aout au 02. September 2005.

Watkins, S and Alam, F, “Future vehicle thermal cooling and aerodynamic drag savings: where will they come from?”, Proceedings of International Conference on Advanced Vehicle Technologies and Integration, China Machine press, pp. 775-82, 17-20 January, Changchun, China, 2012.

“World Energy Outlook”, Executive summary, China and India Insights, International Energy Agency IEA, 2007.