VERIFICATION OF CFD METHOD FOR VEHICLE FLOW SIMULATION

ANG CHAN YONG

A report submitted
In fulfilment of the requirements for the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2021

DECLARATION

I declare that this project entitled "Verification of CFD Method for Vehicle Flow Simulation" is the result of my own work except as cited in the references.

Signature : ....

Name : ANG CHAN YONG.......................

Date : 20 JUNE 2021...............................

APPROVAL

I hereby declare that I have read this project report and in my opinion that this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature : ....

Supervisor's Name : CHENG SEE YUAN

Date : 23/9/21

DEDICATION

I would like to dedicate the success of this project to my families who have given a lot of support and encouragement to complete the project. Second of all, dedication to my fellow friends that have helped me in terms of moral support. I will always remember and appreciate for what they have done. Last but not least, I would also like to express my appreciation to my supervisor, Dr Cheng See Yuan who supervise and guide me along the research. Without his contribution, this report would not be accomplished.

ABSTRACT

Computational Fluid Dynamics (CFD) had been introduced as an engineering tool that was devoted to the solution of fluid flow through computers. However, the question had arisen on how confidence in modelling and simulating be critically accessed. Verification of the CFD approach was conducted to determine the prediction of the results. Ahmed model, which was devised by Ahmed, was designed and simulated via ANSYS fluent. Four verification methods were applied for this research, such as grid independence test, comparison of turbulence models, comparison of time flow and changes of the downstream length of the domain. Each method was labelled as Step, and the drag coefficient from the simulated result was compared with the experimental result conducted by Ahmed in 1984 ($C_D = 0.29$). In Step 1, the grid-independent test showed that the Medium meshing had a percentage error of only 1.88%, which was chosen as it had the closest result to the experimental result. In Step 2, the initial turbulence model, K-epsilon was chosen as it had a nearer result with a percentage error of 1.88%, compared to K-omega with a percentage error of 11.95%. As for Step 3, the comparison between the steady flow and transient flow showed that the result of the transient flow was preferable as it greatly pulled the difference between the simulated result and the experimental result to a minor error of 0.17% only. For the last step, the changes in the downstream length of the domain barely affect the drag and lift coefficient. The drag and lift coefficient shown in the results be consistent, despite the increase of downstream length. The numerical results in this project were determined and the CFD approach on the vehicle flow simulation was verified.

ABSTRAK

Computational Fluid Dynamics (CFD) telah diperkenalkan sebagai alat kejuruteraan yang dikhaskan untuk menyelesaian masalah aliran bendalir melalui komputer. Namun, persoalan timbul mengenai keupayaan CFD dalam pemodelan dan simulasi sama ada dapat diakses secara kritis. Pengesahan pendekatan CFD dilakukan untuk menentukan ramalan hasilnya. Ahmed model yang dicipta oleh Ahmed, direka dan disimulasikan melalui ANSYS Fluent. Empat kaedah pengesahan digunakan untuk penyelidikan ini, seperti ujian kebebasan grid, perbandingan model turbulensi, perbandingan aliran masa dan perubahan panjang hiliran domain. Setiap kaedah dinamakan Langkah, dan pekali seret dari hasil simulasi dibandingkan dengan hasil eksperimen yang dilakukan oleh Ahmed pada tahun 1984 (C_D = 0.29). Langkah 1, ujian bebas grid menunjukkan bahawa meshing Medium mempunyai kesalahan peratusan hanya 1.88% dan dipilih disebabkan hasilnya yang paling dekat dengan hasil eksperimen. Dalam Langkah 2, model turbulensi permulaan, K-epsilon dipilih kerana mempunyai hasil yang lebih dekat dengan ralat peratusan 1.88%, dibandingkan dengan K-omega yang mempunyai ralat peratusan 11.95%. Bagi Langkah 3, perbandingan antara aliran tetap dan aliran sementara menunjukkan bahawa hasil aliran sementara lebih disukai kerana ia mengurangkan perbezaan antara hasil simulasi dan hasil eksperimen kepada kesalahan kecil yang hanya 0.17%. Untuk langkah terakhir, perubahan panjang hiliran domain hampir tidak mempengaruhi pekali seret dan angkat. Walaupun panjang hilir tersebut meningkat, pekali seret dan angkat yang ditunjukkan dalam hasil agar konsisten. Hasil berangka dalam projek ini ditentukan dan pendekatan CFD pada simulasi aliran kenderaan disahkan.

ACKNOWLEGMENTS

First and foremost, I will like to take this opportunity to express my gratitude to my supervisor, Dr. Cheng See Yuan for his continuous support on my research. His essential supervision, guidance and encouragement had helped me a lot throughout the research and writing of the report. The completion of this project will not be so successful without his knowledge and patience on guiding me.

I will also like to thank my friends for lending a hand. Despite having to conduct tests and writing report, they still use some of their occupied time to teach me about the operation of ANSYS software. They also give some ideas on how I should conduct the progress for this project.

Lastly, a special thanks to my family members for their moral support throughout my life. Once again, I am genuinely happy and thankful to everyone sincerely.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CAE	Computer-Aided Engineering
CFD	Computational Fluid Dynamics
PDE	Partial Differential Equation
RANS	Reynold's Averaged Navier-Stokes

LIST OF SYMBOLS

C_D	= Drag coefficient (Dimensionless)
F_D	= Drag force (N)
ρ	= Density of the fluid (kg/m^3)
V	= Speed of the solid body relative to the fluid (m/s)
A	= Cross-sectional area (m^2)
C_L	= Lift coefficient (Dimensionless)
F_L	= Lift force (N)
L	= Length of Ahmed model (m)
φ	= Slant angle (°)
δ	= Percentage error (%)
v_A	= Actual (measured) value
v_E	= Exact value

CHAPTER 1

INTRODUCTION

1.1 Background

Aerodynamics is a branch of fluid dynamics concern with studying the motion of the fluid, particularly when interacting with an object. There are two fundamental principles in aerodynamic, which are drag and lift. Drag is a force that fluid exerts against the solid body in the flow direction. In contrast, the lift is a perpendicular force that can generate lift to create a rising phenomenon on the solid body. In-vehicle perspective, the primary concern of vehicle aerodynamics is the reduction of drag force, which in turn helps to reduce the fuel consumption and stabilize the vehicle when moving at high-speed (Lucas, 2014). Lift force is undesirable for the vehicle, as it can cause the overturned of the vehicle. Therefore, engineers design the vehicle shape in the opposite way the aircraft does to push the vehicle down to the ground (Katz, 2016). Before the computational simulation emerges, most vehicle manufacturers conduct experiments via wind tunnels to determine the performance of the vehicle streamline. However, the operation of the wind tunnels is too complicated and required much amount of expense and time to conduct. Computer-Aided Engineering (CAE) has become a favourite tool for engineers, allowing engineers to evaluate vehicle design. Computational Fluid Dynamics (CFD) is one of the CAE that has been widely used in different aspects (Blazek, 2015)

Computational Fluid Dynamics (CFD) is introduced as an engineering tool that devoted in the solution of fluid flow through computer. CFD was first introduced in the early 1970, where it had become an acronym for the convergence of physics and numerical mathematics in simulating the fluid flow. CFD had been a pioneer in the early stage of prototype construction, where it allows the engineers to analyse, optimize and verify the performance of the designs. Other than that, CFD simulation is the fundamental process to predict the function of the key design elements to minimize risks that would otherwise persist until the later stage, where the design is hardly able to alter. However, the results came from the CFD simulation are not exactly accurate. Means that, the CFD simulation unable to obtain a direct solution, but only an approximate estimation for the solution. Hence, whether CFD results accurately represent the consistent solutions should be determined and verified (Udoewa & Kumar, 2011).

Verification is a process for determining whether the mathematical model is solved correctly, or simply define as 'solving the equations right' (Roache, 1998). Accuracy indicates the closeness of agreement between the simulation values. Therefore, verification can also be used to identify the accuracy of the mathematical models in terms of mathematical approach.

1.2 Problem Statement

Errors and uncertainties can be described as the deficiencies that may happen in any phrase or activities of the modelling process. In most cases, these happen due to the iterative solution methods and the specification of various input parameters. Such example was the mesh generation of the modelling, a fixed design of a model with different meshing may come out with different results. Means that, if the model that been simulated is compared to the same model with different meshing, the results between the simulations will be differ. The same goes for the settings during the pre-processing, where the results of the simulation depends on what the solver code in CFD reads on the setup information that the users set.

In order to determine whether the computational implementation of the model is correct, verification method was utilized. Verification is one of the primary methods to access accuracy and reliability in computational simulation. To find out whether the model implementation accurately represents the developer's conceptual description of the model, the design geometry and the drag coefficient of the Ahmed model from Ahmed (1984) was used as the reference for the test. The simulation was conducted via ANSYS Fluent and the results were compared to the reference result to select the most suitable setup that precisely match the reference result.

1.3 Objective

The objectives of this project are as follows:

1. To determine the behaviour of the vehicle flow by conducting a simulation using computational fluid dynamics (CFD).
2. To identify the suitable setup to be input into the simulation and utilize the setting for the tests.
3. To determine and compare the value of drag coefficient and lift coefficient for each test with different setup parameters.

1.4 Scope of Report

This project focuses on conducting the simulation on a vehicle model. It was simulated through ANSYS Fluent, which consists of computational fluid dynamics (CFD). The Ahmed model represents the vehicle model due to its simplistic design that allows less time-consuming simulation. Different verification methods, such as grid independent test, comparison of turbulence methods, comparison of time flows and changes of downstream length of fluid domain were conducted to verify the CFD approach.

CHAPTER 2

LITERATURE REVIEW

2.1 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the field of science dedicated to the numerical solution of the equations governing fluid studies in a defined flow geometry (Versteeg & Malalasekera, 2007). The development of CFD can be traced back to early 1970, where it became the acronym for the convergence of scientific comprehension to simulate fluid flows. The beginning of CFD was triggered by the advent of computers and evolved along with computer technology advancement (Blazek, 2015).

Before establishing CFD, the experimental and analytical method had been used in the research involving fluid flow. However, these methods were often limited by constraints such as model size, flow field disturbance, and measurement accuracy. The emergence of CFD compensates for the deficiencies of the experimental and analytical methods and offers access for engineers to visualize the experiments (Wei, 2017). CFD had become an essential tool in modern days for analysis purposes and was used to support and complement both experiments and analytics. The reliance on using the CFD approach was significant due to its quick assessment of design variation, which permits the engineers to predict fluid flow behaviours in a short range of time. Till these days, the experimental and analytical methods were still used to analyse problems involving fluid flow and closely interlinked with the CFD method (Anderson & Wendt, 1995).

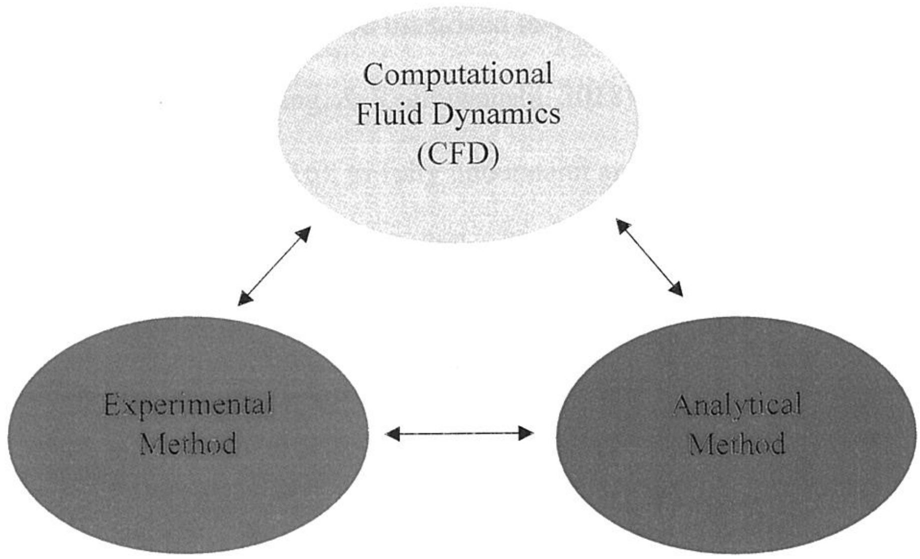

Figure 2.1: Relationship between the basis for fluid studies

Navier-Stokes equations, the fundamental basis for almost all the CFD problems, define the fluid flow in the continuum. The Navier-Stokes equations were first introduced by Clause-Loius Navier in 1822 with viscosity, giving a more realistic yet challenging problem of viscous fluids. George Gabriel Stokes subsequently refined the equations with a more completed solution in 1842 (Hosch, n.d.). These equations were generated based on the conservation law of physical properties of fluid, such as:

- Conservation of mass.
- F = ma (Newton's Second Law).
- Conservation of energy.

In most cases, Navier-Stokes equations are used to define the single-phase fluid flow. Modification or simplification of the equations may have required, depending on fluid flow behaviour in simulation. By excluding specific parameters, Navier-Stokes equations can be simplified to describe viscosity and leads to Euler equations. For further simplification of the equations, parameters involving vorticity can be removed to yield the full potential equations. These equations can then be linearized to yield the linearized potential equations after simplifying (Raman, Dewang, & Raghuwanshi, 2018).

According to Sharma (2016), to fully understand and predict the fluid properties and behaviour, CFD uses the computational method to evaluate models and solve governing equations of fluid dynamics, which are the partial differential equations (PDE). In general, the CFD analysis involved the following steps:

i. **Pre-processing**

The computer model is undergoing a discretization procedure where it is divided into specific discrete points. The assumptions are made as regards the type of flow to be modelled. Other processes to be involved in this phase are mesh generation and application of boundary conditions.

ii. **Solving**

The solving phase covered multiple solvers, varying in efficiency, and capability on solving specific physical phenomena. The solver code reads the setup information from the previous processes, then solves the equations and produces the results files containing the predicted flow properties.

iii. **Post-processing**

For the last step, the obtained results will be visualized, which enable the analyst to verify the results. The obtained results will be presented either in the form of a graph or table.

2.2 Aerodynamics Properties

Aerodynamics is the branch of physics that studies the motion of fluids, particularly when investigating the impact on the solid bodies placed in the flow field. Aerodynamics is a sub-field of the fluid dynamic studies, whereby most of its theories were relevant to the fluid dynamic (Lucas, 2014). The development of aerodynamics was triggered by the desire for flight. Otto Lilienthal was the first to invent the glider flights based on aerodynamics studies. After the research work, the first airplane was successfully created by the Wrights brothers in 1903 (Petrescu et al., 2017). Since then, aerodynamics has acted as the basis for the study of aeronautics and any potential design involving the relevant principles. Engineers apply aerodynamics principles to the design of different bodies, including bridges, buildings, or even sports equipment such as American football. However, the preliminary design based on aerodynamics will be aircraft and vehicles. There are four fundamental forces to be considered in aerodynamics studies, such as lift, thrust, weight, and drag. These forces allow the motion of objects, whether to move up and down and faster or slower. The primarily concerned forces in aerodynamics studies are drag and lift, caused by the air passed through the solid bodies (Lucas, 2014).

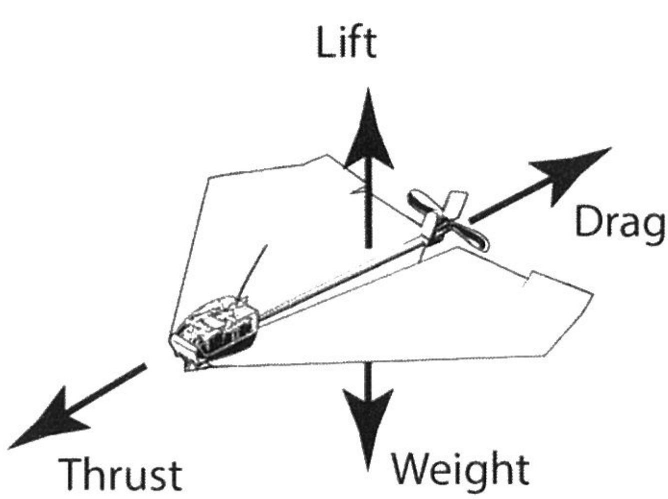

Figure 2.2: Fundamental forces of aerodynamics (Anonymous, 2015)

2.2.1 Drag

When a body is forced to move through a fluid, an interaction between the body and fluid occurs; particularly a liquid, the body will experience some difficulties of forwarding movement due to the resistance the fluid exerted on the body. In short, the force of a flowing fluid exerted on the body in the flow direction is called drag. Drag is an undesirable effect, like friction, was a force that opposes the motion of a body relative to the fluid (Munson, Young, Okiishi, & Huebsch, 2009). Drag is an inevitable phenomenon that occurs whenever there is an object moving through the fluid. In some cases, such as in aircraft and vehicles, drag takes energy to overcome it (Lucas, 2014). Its magnitude and the way to reduce it are significant for vehicles and aircraft engineers as it is closely associated with fuel consumption. However, there are some cases where the drag force is beneficial, such as the brakes of vehicles, for example (Cengel & Cimbala, 2013).

Drag force is conventionally described in terms of a dimensionless quantity called the drag coefficient, define irrespective of the shape of the solid body. The number representing the drag coefficient is dependent not only on the shape of the body but also on other factors, such as the speed, surface roughness, density of the fluid, and the fluid flow was either laminar or turbulent (Lucas, 2014). The drag force depends on various functions, and it is not a practical way to list the force for a variety of situations. Instead, it is more convenient to work with the drag coefficient representing the drag characteristics of the solid body (Cengel & Cimbala, 2013). The drag coefficient can be computed using the following equation:

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A}$$

where

C_D = Drag coefficient (Dimensionless)

F_D = Drag force (N)

ρ = Density of the fluid (kg/m^3)

V = Speed of the solid body relative to the fluid (m/s)

A = Cross-sectional area (m^2)

2.2.2 Lift

To create the rising phenomena of the aircraft, the force exerted on the airfoil must be equals to or exceeds the force of gravity. The force is called lift. In the heavier-than-air craft, the lift was generated by the flow of air over the airfoil. The surface area on the top of the airfoil was more extensive than on the bottom, causing the air to flow over the airfoil to displace more by the top surface, which in turn increases the velocities. According to the Bernoulli equation, an increase in velocity will lead to a decrease in pressure. As the air flows more on the top surface, the pressure loss over the top surface will be greater than that of the bottom surface. Thus, resulted in the arisen of lift force (Connor, 2019).

The vehicle was designed like an airfoil, which will experience a vertical force that lifts the vehicle. However, the lift force was unfavourable for the vehicle. It will reach a point where the gravitational force was negated, and the tip of the vehicle gets lifted up, thus resulting in the overturned of the vehicle. Downforce was a downwards lift force generated to allow the vehicle to travel faster. Unlike the aerodynamics design for the airfoil, the aerodynamics design for the vehicle is preferable to make it low enough. The air flows faster under the vehicle, making the vehicle stick on the ground rather than lift up (Connor, 2019).

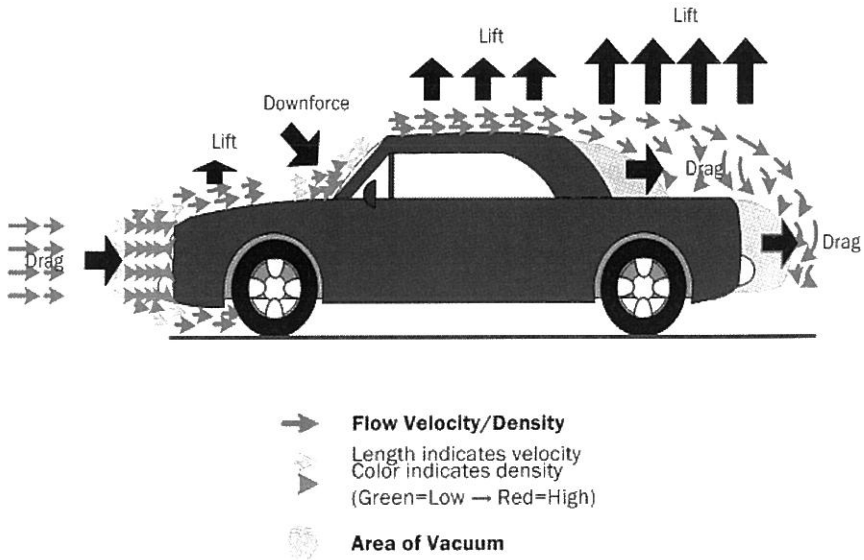

Figure 2.3: Lift and downforce from over body flow (Anonymous, n.d.)

Similar to drag force, lift force was conventionally described by lift coefficient, which represent the lift characteristics of the solid body (Cengel & Cimbala, 2013). The lift coefficient can be computed using the following equation:

$$C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A}$$

where

C_L = Lift coefficient (Dimensionless)

F_L = Lift force (N)

ρ = Density of the fluid (kg/m^3)

V = Speed of the solid body relative to the fluid (m/s)

A = Cross-sectional area (m^2)

2.3 Vehicle Aerodynamics

The first-ever practical vehicle powered by the internal-combustion engine was developed by a German engineer, Carl Benz, in 1885. Although the three-wheeled motor car, also known as 'Motorwagen' ran in 1885, it was patent by its creator in 1886, which marked the birth of the vehicle that will change the history of transportation (Cox, 2017). The innovation of the vehicle replaced the carriage horse, which allows long-distance traveling. During that time, the concept of aerodynamics was not applied to the design of vehicles due to the study of aerodynamic was not fully developed. As the engines became more powerful and the vehicles became faster, the engineers realized that the wind resistance hindered their speed. Not only that, the wind resistance also increased fuel consumption, which was not a favourable condition, especially for that of the gasoline-powered vehicle. The idea of applying aerodynamic to vehicles later emerged after the flight technology had made considerable progress (Lucas, 2014).

In fluid mechanics terms, the vehicle is considered a bluff body that is close to the ground. Typically for a bluff body, the force experienced by the vehicle is mainly pressure drag, which is in contrast to aircraft that suffer primarily from friction drag (Hucho & Sovran, 1993). The consideration for the reduction of drag force leads to the development of aerodynamics vehicles. The initial purpose of studying aerodynamics is to keep the aircraft in the air, but the vehicles, on the other hand, alter the studies to keep it in contact with the ground. This is accomplished by using Bernoulli's effect entirely the opposite way aircraft does. The vehicle is shaped the other way around so that the air deflected underneath it can travel at higher speeds. Therefore, creating a lower pressure area which pushed the vehicle downward (Katz, 2016).