DIMPLE AUGMENTED ON SIDE AHMED BODY: AN ANALYSIS OF TURBULENCE KINETIC ENERGY ON DRAG REDUCTION

MUHAMMAD NORHAIZAD BIN RUKIMAN

B092110060

BACHELOR OF MECHANICAL TECHNOLOGY & ENGINEERING (AUTOMOTIVE TECHNOLOGY) WITH HONORS

2024

FACULTY OF MECHANICAL TECHNOLOGY AND ENGINEERING

DIMPLE AUGMENTED ON SIDE AHMED BODY: AN ANALYSIS OF TURBULENCE KINETIC ENERGY ON DRAG REDUCTION

MUHAMMAD NORHAIZAD BIN RUKIMAN

Bachelor of Mechanical Technology and Engineering

(Automotive Technology) with Honors

2024

DIMPLE AUGMENTED ON SIDE AHMED BODY: AN ANALYSIS OF TURBULENCE KINETIC ENERGY ON DRAG REDUCTION

MUHAMMAD NORHAIZAD BIN RUKIMAN

A thesis submitted

In fulfillment of the requirements for the degree of

Bachelor of Mechanical Engineering and Technology

(Automotive Technology) with Honors

Faculty of Mechanical Technology and Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2024

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **DIMPLE AUGMENTED ON SIDE AHMED BODY: AN ANALYSIS OF TURBULENCE KINETIC ENERGY ON DRAG REDUCTION**

SESI PENGAJIAN: **2024-2025 Semester 1**

Saya **MUHAMMAD NORHAIZAD BIN RUKIMAN**

mengaku membenarkan tesis ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hak milik Universiti Teknikal Malaysia Melaka dan penulis.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja dengan izin penulis.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

☐ TERHAD (Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia sebagaimana yang termaktub dalam AKTA RAHSIA RASMI 1972)

☐ SULIT (Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)

☑ TIDAK TERHAD

Disahkan oleh:

Alamat Tetap:

Cop Rasmi:

Tarikh: 6 JANUARY 2025

Tarikh: 10 JANUARY 2025

** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh laporan PSM ini perlu dikelaskan sebagai SULIT atau TERHAD.

DECLARATION

I declare that this Choose an item entitle "Dimple Augmented Ahmed Body Side Design: An Analysis of Boundary Layer Turbulence and Drag Reduction using Total Kinetic Energy" is the result of my own research except as cited in the references. Choose an item has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature :

Name : MUHAMMAD NORHAIZAD BIN RUKIMAN

Date : 6 JANUARY 2025

APPROVAL

I hereby declare that I have checked this thesis and in my opinion, this thesis is adequate in terms of scope and quality for the award of the Bachelor of Mechanical Technology and Engineering (Automotive Technology) with Honors.

Signature :

Supervisor Name : TS. MOHD FARUQ BIN ABDUL LATIF

Date : 10 JANUARY 2025

DEDICATION

This thesis are dedicated to:

The sake of Allah, My creator and My Master.

My beloved parents, Rukiman Bin Saidin and Norlela Binti Ismail. Your endless love, support, and sacrifices have been the foundation of my journey. You have instilled in me the values of hard work, perseverance, and integrity, and for that, I am eternally grateful.

To my esteemed supervisor, Ts Mohd Faruq bin Abdul Latif, your guidance and mentorship have been pivotal in the completion of this project. Your dedication to excellence has inspired me to strive for the best in my work.

I also dedicate this work to Universiti Teknikal Melaka Malaysia (UTeM) and the Faculty of Mechanical Technology and Engineering (FTKM). The knowledge and opportunities provided by this esteemed institution have been instrumental in my academic and personal growth.

Lastly, to all my friends and colleagues who have supported and encouraged me throughout this journey, thank you for your unwavering belief in me. This achievement is as much yours as it is mine.

ABSTRACT

Aerodynamics forms an important part of fluid mechanics, especially in enhancing vehicle performance. The Ahmed body model, formulated in 1984, is a fundamental tool employed in automotive aerodynamic studies. This model has provided great insights regarding drag and wake phenomena, yet its geometry is too simple to be used in practical applications. Maximum drag reduction with the Ahmed body remains very challenging, especially with the application of passive control methods using surface modification. More research is needed to entirely understand the exact effects of dimples on the aerodynamics of the Ahmed body, as dimples can alter boundary layer behavior and thus reduce drag. This study will aim to analyze how dimple augmentation of the lateral surfaces of the Ahmed body will affect: turbulence kinetic energy (TKE) and hence drag reduction, changes in boundary layer behavior and wake characteristics, differences in the aerodynamic performance of a dimple-augmented Ahmed body with that of the conventional smooth surfaced Ahmed body. The study will try to enhance the Ahmed body model by the incorporation of different dimple configurations. This shall be achieved by coupling Computational Fluid Dynamics (CFD) simulations with experimental validation. High-resolution meshing will be used and advanced turbulence modeling employed in the simulation of flow patterns and aerodynamic forces. Wind tunnel experiments shall be conducted to validate the CFD results. It is expected that the augmentation of the dimples will enhance the aerodynamic performance of the Ahmed body by leading to a reduction in drag and an alteration in the distribution of TKE, causing the boundary layer to adhere better, reducing turbulence in the wake, leading to the lowering of total drag and enhancing the efficiency of the vehicle. These discoveries have the potential to dramatically enhance the development of vehicle design with improved fuel efficiency and a greater focus on environmental sustainability.

ABSTRAK

Aerodinamik merupakan bahagian penting dalam mekanik cecair, terutamanya dalam meningkatkan prestasi kenderaan. Ahmed Body, dibentuk pada tahun 1984, merupakan alat asas yang digunakan dalam kajian aerodinamik automotif. Model ini telah memberikan wawasan yang baik mengenai fenomena rintangan, tetapi geometrinya terlalu mudah untuk digunakan dalam aplikasi praktikal. Pengurangan tarikan maksimum dengan Ahmed Body kekal sangat mencabar, terutamanya dengan penggunaan kaedah kawalan pasif menggunakan modifikasi permukaan. Lebih banyak penyelidikan diperlukan untuk sepenuhnya memahami kesan yang tepat daripada dimples pada aerodinamik Ahmed Body, kerana dimples boleh mengubah tingkah laku lapisan sempadan dan dengan itu mengurangkan tarikan. Kajian ini akan bertujuan untuk menganalisis bagaimana peningkatan dimple permukaan lateral Ahmed Body akan mempengaruhi: tenaga kinetik turbulensi (TKE) dan oleh itu pengurangan rintangan, perubahan dalam tingkah laku lapisan sempadan dan ciri-ciri bangun, perbezaan dalam prestasi aerodinamik Ahmed Body dimple-meningkatkan dengan permukaan halus konvensional. Kajian ini akan cuba meningkatkan model Ahmed Body dengan menggabungkan konfigurasi dimple yang berbeza. Ini akan dicapai dengan menggabungkan simulasi CFD (Computational Fluid Dynamics) dengan pengesahan eksperimen. Meshing resolusi tinggi akan digunakan dan pemodelan turbulensi canggih digunakan dalam simulasi corak aliran dan kekuatan aerodinamik. Ujian terowongan angin akan dijalankan untuk mengesahkan hasil CFD. Ia dijangka bahawa peningkatan dimples akan meningkatkan prestasi aerodinamik Ahmed Body dengan membawa kepada pengurangan rintangan dan perubahan dalam pengedaran TKE, menyebabkan lapisan sempadan untuk menempel lebih baik, mengurangkan turbulensi di belakang, yang membawa kepada penurunan jumlah rintangan dan meningkatkan kecekapan kenderaan. Penemuan-penemuan ini mempunyai potensi untuk secara drastik meningkatkan pembangunan reka bentuk kenderaan dengan peningkatan kecekapan bahan api dan tumpuan yang lebih besar kepada kesinambungan alam sekitar.

ACKNOWLEDGEMENTS

In the Name of Allah, the Most Gracious, the Most Merciful

I would like to thank my supervisor, Ts Mohd Faruq bin Abdul Latif, whose guidance, support, and motivation were invaluable throughout this project. He has rendered his expertise and input in shaping this work. I am sincerely grateful for the patience he shows and for mentoring me.

I would like to thank Universiti Teknikal Melaka Malaysia (UTeM) and also the Faculty of Mechanical Technology and Engineering (FTKM) for providing the resources needed for my research and creating a conducive environment. Support from faculty members and access to facilities at par with world standards have been prime reasons for the conduct and successful completion of this study.

My heartfelt thanks are extended to my parents, Rukiman Bin Saidin and Norlela Binti Ismail, for their unwavering support, love, and understanding. Their encouragement and firm belief in my abilities have been a constant source of motivation. Without their sacrifices and constant encouragement, this achievement would not have been possible.

Finally, I would like to thank all my friends and colleagues who have supported me in various ways during this journey. Your support and encouragement have been appreciated.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS AND ABBREVIATIONS

CFD	-	Computational Fluid Dynamic
TKE	-	Total Kinetic Energy
RANS	-	Reynolds-Averaged Navier-Stokes
Cd	-	Coefficient of Drag
Cp	-	Coefficient of Pressure
PANS	-	Partially Averaged Navier-Stokes
k-ϵ	-	K – Epsilon Turbulence Model
k-ω	-	K – Omega Turbulence Model
LES	-	Large – Eddy Simulations
SST	-	Shear Stress Transport
SGS	-	Sub Grid-Scale
DES	-	Detached Eddy Simulation
Re	-	Reynolds Number
AHTM	-	Arbitrary Hybrid Turbulence Modeling
EARSM	-	Explicit Algebraic Reynolds Stress Model
Φ	-	Angle
R	-	Radius
L	-	Length
H	-	Height
W	-	Width
ρ	-	Fluid density
U	-	Velocity
μ	-	Dynamic Viscosity.

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1. Background of the study

Aerodynamics is one of the main branches of fluid mechanics and is crucial in the improvement of vehicle performance. In particular, the Ahmed body, developed by S. R. Ahmed in 1984, is one of the most important models in automotive aerodynamics research (Ahmed et al., n.d.). This simple model, which replicates the rear shape of a car and is matched with certain proportions, allows the detailed study of the generation of aerodynamic phenomena such as drag and wake. Scientists have exploited it to understand the influence of rear slant angles of different rear slant angles on airflow, providing vital knowledge in vehicle aerodynamics and enhancing the techniques to be used in the design of vehicles that deliver less fuel consumption and emissions.

While the Ahmed body has greatly developed our knowledge of aerodynamics, it is still important to recognize that this model is imperfect, and the results from wind tunnel tests are not always a perfect indication of what happens in reality. This study focuses on improving the model at present by integrating actual car characteristics to further increase the level of detail and, on the other hand, combining empirical data with computational simulations. (Ahmed et al., n.d.) One of the most successful methods is dimple augmentation of the Ahmed body to add dimples on the surface, similar to those on golf balls, to enhance drag reduction by affecting the behavior of the boundary layer and the turbulence. (Mohammadikalakoo et al., 2020) This study aims to analyze the effect of adding dimples on the sides of the Ahmed body at strategic positions to estimate their potential to decrease drag and enhance the performance of the vehicle.

Understanding the complete phenomenon of boundary layer turbulence is essential for the optimal aerodynamics of the vehicle. Boundary layer turbulence is the unsteady and changing behavior of velocity and pressure near a surface. This leads to drag force and transmission of heat. (Lehmkuhl et al., 2019) Advances in computer fluid dynamics have helped to simulate these complex phenomena. Researchers have studied several drag reduction techniques like vortex generators and surface modifications. (Essel et al., 2020) This paper aims to study comprehensively the augmentation of dimples on Ahmed body. To be specific, the reduction of drag will be measured, and a correlation between turbulent kinetic energy and the decrease in drag will be established. These findings will aid in constructing a more aerodynamic car, which will enhance fuel efficiency, safety, and environmental protection.

1.2. Problem statement

The Ahmed body is still a widely used model for automobile aerodynamics research as it is an inexpensive model and provides a simple yet accurate representation of real-life vehicle forms. This model allows research to study various airflow phenomena, which include separation, reattachment and wake creation, which play an essential role in understanding the drag forces acting on vehicles. Since the advent of the Ahmed model in 1984, and numerous empirical and computational investigations, finding an optimal drag minimization of the Ahmed body in an automobile is still a challenging task, especially concerning drag minimization using passive control methods. (Ahmed et al., n.d.)

Passive flow control methods, providing aerodynamic benefits without the use of outside energy input, have been studied as a result. Among these methods, surface changes, including dimples, vortex generators, and riblets have been studied. Dimple's ability to delay flow separation and increase turbulent flow over a surface has shown potential for passive drag reduction. The potential for such surface modifications, in various aerodynamic scenarios, has been demonstrated. (Koppa Shivanna et al., 2021; Viswanathan, 2021). However, the effect of dimples on the aerodynamic performance of the Ahmed body requires more study.

To fill these gaps, the present study would like to investigate the effects of dimple augmentation on the lateral surfaces of an Ahmed body. The study will focus on TKE and drag minimization. Given this, the study will aim to determine the changes in turbulent kinetic energy (TKE) and drag coefficients that have resulted from adjustments in the dimple configuration on the surface of the test model, using the latest CFD methods. On the other hand, this paper will discuss the dimensions, layout, and placement of the dimples that lead to the most positive aerodynamic benefits. The understanding and improvement of the aerodynamic performance of vehicles, including the Ahmed body, is at the core of fuel efficiency, emission reduction, and vehicle stability and goes hand in hand with these parameters. It is not only an assurance of safety for the environment but also a value for the creation of economic value for the automobile industry. (Akhter et al., 2022; Luo et al., 2022).

1.3 Objective

The present study aims to comprehensively examine and comprehend the effects of dimple augmentation on the lateral surfaces of an Ahmed body with a particular emphasis on:

i. To analyze the effect of dimple augmentation on the side surfaces of the Ahmed body on turbulence kinetic energy on drag reduction.

ii. To validate the analysis using Grid Sensitivity Analysis

iii. To evaluate the relationship between turbulence kinetic energy with drag reduction

1.4. Project Scope

This project will be strongly focused on the enhancement of the aerodynamic performance of the Ahmed body by strategic dimple application. Details of the investigation using state-of-the-art CFD and complete flow visualization techniques are within the scope. The specific areas are identified below:

1.4.1. Ahmed Body Model

- Baseline Geometry:

The Ahmed body model is a well-recognized simplified geometric representation used in vehicle aerodynamic studies. The use of this in the project allows consistency with earlier research to produce a reliable benchmark for comparison. Using this standard model, existing data and methodologies are employed, and any observed effects can be guaranteed to be due to the modifications by dimples and not to some inherent properties of the body itself.

- Dimple Integration:

This part of the work is focused on the integration of dimples into the Ahmed body. It involves the exploration of different sizes, shapes, and patterns of dimple distributions. Such an investigation is important because the efficacy of dimples in drag reduction may well depend strongly on their geometries. Any systematic change to such parameters will yield the most efficient designs and thus inform about the relationship between surface modifications and aerodynamic performance.

1.4.2. Computational Fluid Dynamics

- Analysis Simulation Framework:

An effective CFD framework is to be created to simulate complex flow patterns existing around both the regular and dimpled Ahmed bodies effectively. Very high-resolution meshing will be necessary to represent the dimple geometries' subtleties and related flow interactions. The CFD simulations will provide a detailed explanation of the flow patterns, pressure distribution, and the resulting aerodynamic forces, which are quite difficult to assess through testing.

- Turbulence Modeling:

Turbulence modeling is the simulation and mathematical description of turbulent flow events. To correctly feel the effect of dimples on the behavior of the boundary layer and flow separation, the application of contemporary turbulence models like RANS and appropriate near-wall treatments is essential. The knowledge of the way dimples influence the flow field, and, particularly how they may generate favorable turbulence that could be beneficial.

1.4.3. Visualization and Analysis of Flows

- Boundary Layer Investigation:

Flow visualization methods, such as turbulence visualization and surface visualization, are to be used in the comprehensive investigation of boundary layer behavior. These techniques will provide qualitative and quantitative information regarding the effects of dimples on separation points, flow reattachment, and turbulence generation. To optimize dimple designs, these factors must be well understood.

- Experimental Validation:

Wind tunnel tests have to be included in CFD models for experimental validation of the computed results. Verification of simulation results with accurately measured experimental data can help ensure that the reported effects are not artifacts of the numerical approaches but indeed represent actual physical occurrences.

1.5. Expected Result

The study is expected to yield useful information regarding how the augmentation of dimples influences the aerodynamics of Ahmed body, with the main focus resting on three main objectives. First, the dimples are expected to have an overall effect on the turbulence kinetic energy (TKE) by modifying its distribution around the Ahmed body. Localized spikes in the values of TKE are likely to appear, especially in the vicinity of the dimple region, which can be considered as a positive parameter for improved mixing in the boundary layer and energy dissipation. This is expected to result in a decrease, on average, in the TKE levels, especially in the wake region of the body, because of more controlled and ordered flow with the presence of the dimples.

From the literature review, it is anticipated that dimples are likely to yield their impact in decreasing the aerodynamic drag by a considerably large factor in terms of a decrease in the drag coefficient (Cd) and, thus, in the overall drag force on the Ahmed body with dimples. This decreased aerodynamic resistance that the surface with dimples will experience is analyzed experimentally and through simulation-based methods as well, with the results showing evidence of its effective operation.

Next, it is anticipated that dimples will cause an alteration in the flow properties and behavior of the boundary layer, proving to have a better attachment of flow as compared to that without dimples. This is expected to result in a delay in the separation of flow and an earlier attachment on the body sides of the Ahmed body, leading to a more stable and strongly attached boundary layer, and hence less pressure drag. Also, the dimples are expected to decrease the scale and intensity of the vortices being shed in the wake, hence presenting a better-streamlined wake region. In the boundary layer behavior, an in-depth study is expected to show that dimples manage a thinner and stronger boundary layer, hence showing decreased skin friction and total drag. These will be proof enough for the fact that dimple augmentation can indeed cause aerodynamic improvement, hence a strong base for further research and highly concrete practical applications in the design of automobiles.

CHAPTER 2

LITERATURE REVIEW

2.1. Introduction

Aerodynamics is one of the main branches of fluid mechanics and is crucial in the improvement of vehicle performance. In particular, the Ahmed body, developed by S. R. Ahmed in 1984, is one of the most important models in automotive aerodynamics research (Ahmed et al., n.d.). This simple model, which replicates the rear shape of a car and is matched with certain proportions, allows the detailed study of the generation of aerodynamic phenomena such as drag and wake. Scientists have exploited it to understand the influence of rear slant angles of different rear slant angles on airflow, providing vital knowledge in vehicle aerodynamics and enhancing the techniques to be used in the design of vehicles that deliver less fuel consumption and emissions.

While the Ahmed body has greatly developed our knowledge of aerodynamics, it is still important to recognize that this model is imperfect, and the results from wind tunnel tests are not always a perfect indication of what happens in reality. This study focuses on improving the model at present by integrating actual car characteristics to further increase the level of detail and, on the other hand, combining empirical data with computational simulations. (Ahmed et al., n.d.) One of the most successful methods is dimple augmentation of the Ahmed body to add dimples on the surface, similar to those on golf balls, to enhance drag reduction by affecting the behaviour of the boundary layer and the turbulence. (Mohammadikalakoo et al., 2020)This study aims to analyse the effect of adding dimples on the sides of the Ahmed body at strategic positions to estimate their potential to decrease drag and enhance the performance of the vehicle.

Understanding the complete phenomenon of boundary layer turbulence is essential for the optimal aerodynamics of the vehicle. Boundary layer turbulence is the unsteady and changing behaviour of velocity and pressure near a surface. This leads to drag force and transmission of heat. (Lehmkuhl et al., 2019) Advances in computer fluid dynamics have helped to simulate these complex phenomena. Researchers have studied several drag reduction techniques like vortex generators and surface modifications. To be specific, the reduction of drag will be measured, and a correlation between turbulent kinetic energy and the decrease in drag will be established. These findings will aid in constructing a more aerodynamic car, which will enhance fuel efficiency, safety, and environmental protection.

2.2. Systematic Literature Review

The proposed strategy is a method-stimulated method for doing a scientific literature overview of a specific area. The methodological framework, inside the form of planning, execution, and reporting, is the guiding mild for researchers embarking on an exploration through the maze of scholarly literature. Although the method offers a systematic manner from formulating the research questions (Tranfield et al., n.d.) "My," to a country of expertise (*Systematic Reviews In The Social Sciences_ A Practical Guide By Mark Petticrew*, n.d.), "The," a vital evaluation outlines a few limitations and challenges for consideration (Greenhalgh et al., 2018).

During the preliminary degree of research, study questions and the development of a conceptual framework are the foci—an exercise in comparison to carving a "ventifact" out of uncooked thoughts. (Arksey & O'Malley, 2005) While this approach brings greater distinctiveness to the method and helps in building seek criteria, it also infiltrates a stage of subjectivity. Since idea generation is a subjective technique, it can subtly adjust the direction of the research and the choice and interpretations of the literature. (Webster & Watson, 2002) Further, incorporating a case look at by myself for the illustration of a practical software of the method may tend to restrict the scope of the findings and forget a couple of variants of the research context, as a result restricting the generalizability of the technique (Tranfield et al., n.d.).

Finally, although the technique indicates an extensive representation of the applicable literature and profound analysis, it's far incumbent upon the researcher to pay awesome care whilst executing the process. In this regard, the selection of inclusion and exclusion criteria and their execution rely on the cautious choice of these paintings. (*Systematic Reviews In The Social Sciences_ A Practical Guide By Mark Petticrew*, n.d.) Due to the absence of pointers and standardized tactics, an opportunity for inconsistency or vagueness in the technique of selection of literature may additionally ensue. (Greenhalgh et al., 2018) Thus, researchers ought to work out brilliant care and excessive control tactics for a hit behavior of their literature evaluation manner. (Arksey & O'Malley, 2005)

Specifically, this proposed technique offers a hard and fast method on the way to cope with the complexities of the literature review. However, it should be talked about and guarded towards the limitations imposed by the proposed method. Only when every level of the method, coupled with first-class control measures, is nicely evaluated, shall the researchers be capable of enhancing the reliability and robustness of their literature evaluation, and a solid ground shall be laid for additional studies work to be constructed upon. (Webster & Watson, 2002)

2.2.1. Identify keyword

This indeed is a founding step in building effective search strategies for literature reviews by identifying relevant keywords in the titles of current research (Luciano, 2011). Focusing on specific terms such as "Ahmed Body," "Boundary Layer Turbulence," "CFD," and "Total Kinetic Energy" is a good way for scholars to begin their search of the scholarly environment, trying to find studies relevant to their research questions (Webster & Watson, 2002). However, this methodological approach shall, in turn, come with its inherent constraints to ensure a thorough search of the literature.

First, one issue with keyword search is the possible omission of other relevant research that might be expressed in different terms or with different wordings than the chosen keywords. This limitation is a strong reason why this method of searching for titles is not a sole criterion for literature selection and that such searches should be completed with other search criteria, such as the search by abstracts or topic headings (Luciano, 2011). Without this complementarity, the representation of the existing literature will be skewed, and otherwise, unique contributions may be left undiscovered.

Additionally, this bias in the search toward titles containing keywords may introduce bias into the entire process of the literature review, not only for the sake of impartiality but also for a comprehensive approach (Pearson, 2014). Research papers with catchy and fashionable keywords might be drawn disproportionately from the rest, sometimes even at the expense of equally important contributions with less attractive titles. Such an occurrence, besides being a possible distortion of the perspective of the research landscape, is also a demise of the integrity of the literature review as it places visibility above scholarly merit. Given these difficulties, a researcher should be selective and supplement their keyword search with other methods of search to ensure a thorough search of the literature and, consequently, increase its objectivity.

In this context, I recommend that researchers adopt a more holistic search approach that utilizes the combination of diversified search criteria and approaches (Pearson, 2014). When a researcher uses a more multi-faceted approach in his or her research, he or she enhances the rigor and scope of the literature review, thus establishing a firm foundation from which to construct insightful research outcomes. Hence, a holistic approach, of course, not only reduces the possibility of losing precious contributions but also allows one to obtain a more comprehensive grasp of the research terrain and, therefore, contribute more significantly to the scholarly discourse.

2.2.2. Synonym of keyword

The use of thesaurus resources through an online search is a good strategy when performing a literature review (Webster & Watson, 2002). Synonyms extend the vocabulary used in the search to identify crucial terms such as "Ahmed Body," "Drag Reduction," "Boundary Layer Turbulence," and "Total Kinetic Energy," among others(Luciano, 2011). The practice includes more related work in the literature review. The practice increases the scope of the literature search and allows the researcher to understand the subject matter fully by including more points of view and perspectives (Pearson, 2014) as shown in Table 2.2.2.

However, there are some challenges and drawbacks that are faced when synonymizing keywords. The first and most crucial among them is that the practice of using synonyms may cause a language barrier, whereby the search may bring a different outcome if the context of the language is different (Webster & Watson, 2002). For example, there might be a difference in a particular discipline from those found in another region. Therefore, there are chances that the search results may be different from each other. In addition, the use of indiscriminate synonyms without proper checking may result in irrelevant or unrelated material that lowers the focus and logical coherence of the literature review.

Table 2.2.2: Synonym of Keyword

No.	Keyword	Synonym
1	Dimple	Hollow ; Concavity
2	Device	Apparatus ; Mechanism ; Equipment
3	Augmented	Widened ; Expanded
4	Boundary Layer	Barrier
5	Turbulance	Disturbance ; Instability
6	Drag	Nuisance
7	Reduction	Contraction ; Devaluation
8	Kinetic Energy	Driving Force; Electromotive Force

In the context of the study under discussion "Dimple-Augmented: Ahmed Body Side Design," synonymizing keywords brings a broader scope of inquiry on the subject (Pearson, 2014). However, to check such challenges that have been involved with this practice, researchers have to be very discreet in their identification and use of synonyms that are closely related to the stated scope and aims of the review. In this way, the researchers can increase the scope and comprehensiveness of the literature review. The researchers can therefore provide highly sound and related results of studies.

In summary, while there is power in an expansive breadth of the literature search with synonyms of keywords, researchers should take care and exercise caution (Luciano, 2011). It is achieved by taking into consideration the challenges and by carefully exercising thoughtful discretion in choosing synonyms, thereby making the approach effective and adding depth and comprehensiveness to a literature review.

2.2.3. Search String

Systematic literature reviews are largely dependent on the creation of precise search terms based on the use of extensive keyword variants (Webster & Watson, 2002). The search strings made for the Scopus and Web of Science databases suggest a meticulous selection of synonymous terms and related concepts based on the research topic: "Dimple-Augmented: Ahmed Body Side Design: An Analysis of Boundary Layer Turbulence and Drag Reduction using CFD and Total Kinetic Energy on Variable Velocity." (Luciano, 2011). These aspects of the study are addressed by phrases such as "Turbulence," "Disturbance," "Instability," "Kinetic Energy," "TKE," "Ahmed Body," "Drag," and "Reduction." However, these search strings point to a dedicated effort to include as much literature as possible; however, some other subtleties and constraints need to be considered.

Utilizing synonymous phrases and related themes enhances the thoroughness of a search, thus allowing the retrieval of a wide range of relevant research (Pearson, 2014). This recognizes the complexity and heterogeneity of the subject of the study and adjusts to differences in the use of terms that are common in most academic disciplines and sub-disciplines. Utilizing alternative terms such as "Contraction," "Devaluation," and "Driving

Force" allows the search queries to take care of the natural variance in terms used by researchers, thereby minimizing the chances of misleading relevant material due to differences in meaning.

However, despite the power of the search terms, some issues will always militate against the use of synonyms as found in the literature searches. Language differences, although assumed to make sure that a wide range of literature is fetched, may also interfere with the search results and yield not only irrelevant research but peripheral ones as well. Adding closely related terms, such as "Electromotive Force" to the Web of Science search string, along with "Kinetic Energy," may inadvertently reduce the relevancy of the returned material to the subject matter by extending the search beyond that which is targeted. Furthermore, complexity in the search strings may involve problems in the proper parsing and understanding of the search results therefore impeding the synthesis and analysis of findings throughout the review process.

Secondly, the level of detail of certain terms used in the search strings, for instance, "Ahmed Body," may indeed affect the scope of the literature that is garnered (Luciano, 2011). Although the word "Ahmed Body" precisely identifies the subject of the study, its specificity may mistakenly narrow down the search for publications that specifically contain the term "Ahmed Body" in their titles or abstracts. This may also exclude relevant material that discusses similar subjects or events but which do not use the term "Ahmed Body" designation. As a result, researchers must be circumspect in adjusting the balance between specificity and generality when building search terms so that the retrieval of relevant material can be comprehensive and representative. As shown in Table 1, both are search strings with search engines to identify literature reviews.

Table 2.2.3: Search String method

No.	Search Engine	Search String	Result
1	Web of Science	TS = ((Turbulence OR Disturbance OR Instability)&(Drag OR Nuisance)&(Reduction OR Contraction OR Devaluation)&("Kinetic Energy" OR "Driving Force" OR "Electromotive Force" OR TKE))	293
2	Scopus	ALL ((Turbulence OR Disturbance OR Instability) & (Kinetic Energy" OR TKE) & ("Ahmed Body"))	106

Overall, the search strings developed for the Scopus and Web of Science databases serve to make a praiseworthy attempt to incorporate most of the relevant literature about this study subject. However, they also clearly portray the intricate and subtle challenges brought about by doing systematic literature research. Researchers can better enhance their literature exploration by taking critical consideration of the advantages and drawbacks of using different search techniques (Pearson, 2014). This enables them to fine-tune their approach and find a balance between being comprehensive and being specialized to maximize the retrieval of relevant scholarly works.

2.2.4. Preferred Reporting Items for Systematic Reviews and Meta-analysis (PRISMA).

The PRISMA flow diagram is perhaps one of the fundamental tools in systematic reviews and allows for a transparent overview of the literature-selection process (Page et al., 2021). The structured format of the diagram helps readers see the transparent and systematic selection of studies of a systematic review. It is therefore of great value in augmenting the credibility and reproducibility of the research (Moher et al., 2009). The PRISMA flow diagram, though, is not devoid of limitations that require careful consideration. One of the most glaring limitations of the PRISMA flow diagram is the potential for reporting bias to affect the review outcomes (Page et al., 2021). The diagram summarizes the studies included and excluded at each stage, though it does not remove publication bias or selective reporting tendencies in the literature. This may give the impression that certain study categories or results are overrepresented, perhaps skewing the synthesis of evidence presented in the review (Moher et al., 2009).

The PRISMA flow diagram's rigidity may also pose challenges in accommodating the complexities of certain review procedures or research inquiries, especially about qualitative research and mixed-methods approaches (Tricco et al., 2018). The linear format of the diagram may not capture the iterative and reflexive nature that data collection and analysis take in such methods (Shea et al., 2017). It is thus important that the review methodology be flexible enough so that the PRISMA flow diagram can accurately capture the complexities of the research process.

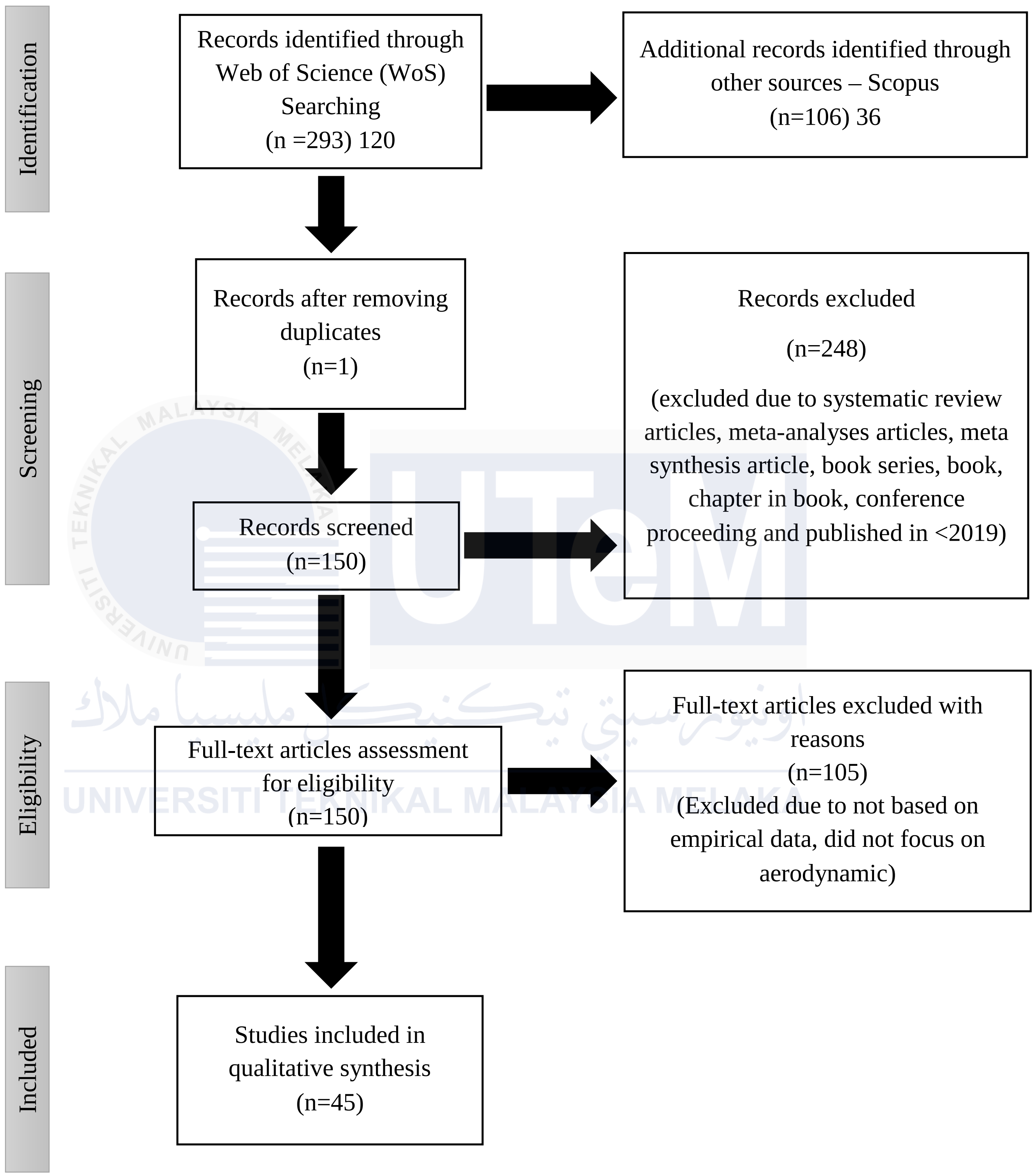


Figure 2.2.4: PRISMA Flow Diagram

With these considerations in light, one would do well to use the PRISMA flow diagram with a critical eye and realize some of its strengths and limitations. Supplementary metrics or strategies can be adopted to ensure a comprehensive synthesis of the literature (Liberati et al., n.d.), in a bid to address potential reporting bias. As such, an acknowledgment of these challenges on the part of the researchers allows a better optimization of the effectiveness of the PRISMA flow diagram, in using it to optimize one of the flow processes of systematic reviews, thereby enhancing the credibility and reliability of findings. (Page et al., 2021)

2.2.4.1. Identification

The first phase of identification—which arrived at many entries from Web of Science and Scopus—is commendable to the extent that it indicates that the search strategy used is thorough. This is commendable because it ensures a wide spectrum of material is considered and means that the scope and depth of review are much wider. However, there is no reporting on the search methodology that is followed, which presents many obstacles to transparency and reproducibility in the systematic review process.

The reporting of search methods is crucial in reproducing systematic reviews and enhancing the readers' ability to know the quality of the review findings. The absence of a clear indication of search terms used, databases consulted, and filters introduced removes all the chances of knowing how extensive the search strategy is, and readers cannot tell what can be detected in terms of biases while choosing the studies. Furthermore, a lack of information makes it impossible to produce a repeatable process of searching, which is critical in checking and confirming the results of the review.

To render the results of the review more credible, as well as transparency of the research process itself, the details of the search methodology adopted should be given in detail. This will involve mentioning the actual keywords or search strings used, the descriptions of the databases that have been consulted, and the mention of the inclusion or exclusion criteria applied. This will give the indication required for readers to assess the effectiveness and impartiality of the search strategy employed.

Transparent reporting about the search methodology improves not only the credibility of a review but also contributes to the development of knowledge in the field. Researchers learn from how other people conducted their searches, refining their approaches and enhancing the collective understanding of effective search strategies. Thus, reporting should be transparent regarding the search methods to foster an open culture and rigor in systematic reviews, which enriches the quality and reliability of findings.

2.2.4.2. Screening

The large number of records that were excluded in the screening phase further points out the reasons for the need for stringent screening criteria to handle the vast amount of literature first identified in the search. The significant number of exclusions also raises concerns over the accuracy and efficacy of the screening methods applied.

There is a risk of subjectivity in the assessment of inclusion/exclusion criteria. In the absence of clearly defined standards to determine inclusion or exclusion, there is also a risk of variable interpretation of screening criteria by reviewers. These may lead to biases or inconsistencies in the decision-making process, compromising the reliability and validity of the review outcome. Hence, a very important aspect is the development of clear and transparent criteria for screening to improve the dependability and accuracy of the review process.

Transparency in reporting screening criteria is important for reproducibility and for the possibility of the readers to assess the rigor of the review methodology. When researchers provide clear details of the criteria used for the determination of study eligibility, they increase transparency and hold themselves accountable for the screening process. Moreover, open reporting of the screening criteria allows for peer assessment and validation, increasing the credibility of the review findings.

Standardized tools or guidelines for screening, such as the PRISMA checklist, may increase consistency and objectivity in the screening process. Such tools provide a framework for reviewers to systematically assess the eligibility of studies, limiting the potential for subjective biases and increasing the reliability of decisions relating to study selection.

So, even though the large number of exclusions during screening clearly shows that strong screening criteria are necessary, it also underscores the importance of transparency and standardization in the screening process. Delineation of clear and transparent standards for screening may increase the dependability and accuracy of systematic review procedures, increasing the credibility and trustworthiness of the review findings.

2.2.4.3. Eligibility

The extensive review of 150 papers shows evidence of a careful reviewing process aimed at the determination of the suitability of studies for inclusion in the review. The very high number of publications excluded due to general irrelevance to aerodynamics leads to questions about the appropriateness of the screening criteria applied initially. This underlines the need to consider what the definition of a subject study requires when developing the research inclusion and exclusion criteria.

A possible reason for such a high number of publications excluded due to general irrelevance to aerodynamics could be the general lack of precision or specificity in the definition of the study subjects, especially about the aerodynamics aspect. The lack of a clear and detailed description of the specific aspects related to aerodynamics of interest for this review meant the reviewers had difficulty assessing the exact relevance of each paper. Such a situation led to the exclusion of important papers that did not clearly describe their focus on aerodynamics.

To make the eligibility evaluation process more efficient and to reduce the number of irrelevant articles detected during screening, the process requires a more precise and detailed description of the study subject. This might include a clear description of key concepts, variables, or phenomena related to aerodynamics of interest. The specificity of the directions of the process increases clarity in the screening criteria, thus making it easier for reviewers to know what types of studies are relevant for inclusion.

Secondly, although there is no mention of either a screening checklist or a framework for the review, the inclusion and exclusion process might be best served by the use of such frameworks, such as the PICOS criteria. This structured approach makes it easier to precisely define the research question and specify the inclusion criteria, thus reducing ambivalence and providing greater reliability in the systematic review process.

Overall, the thorough examination of the papers gives the impression that the selection of the studies is done meticulously. The high exclusion rate suggests the need for reviewing the screening criteria. The offering of more descriptive characteristics of the subject of a study and the use of structured frameworks of study selection will improve the efficiency and effectiveness of the screening for systematic reviews.

2.2.4.4. Included

The fact that 45 papers are present in the qualitative synthesis is an important indicator of a very serious and comprehensive selection of relevant literature for analysis. This indicates that the researchers were very careful in finding and incorporating relevant studies that inform the review. However, the fact that the criteria used to determine research inclusion were not explicitly stated makes it difficult to know how representative and comprehensive the included studies may be. Not having explicit and specific inclusion criteria makes it impossible to know how representative the review is.

Reporting inclusion criteria would have made the systematic review reproducible and reliable. By transparently documenting the inclusion criteria, readers would be in a better position to compare and assess the results of the review in a more meaningful and valid way. Also, the lack of transparency in inclusion criteria introduces ambiguity and subjectivity in the study selection process, potentially leading to inconsistencies in the decisions of including or excluding studies. Inclusion criteria should be clear and specific to improve the transparency and reliability of the review results.

While inclusion criteria have to be reported, other steps need to be undertaken to evaluate and document the presence of biases arising in the selection of studies. Tools, such as the Cochrane Risk of Bias Tool, could be used to objectively and systematically assess the presence of bias in individual studies or across the review. By critically and systematically assessing and documenting the presence of biases, researchers could make readers appreciate the strengths and weaknesses of the review results, thus making the review more credible and trustworthy.

In summary, the attempt to include 95 papers in the qualitative synthesis is commendable but lacks transparency, repeatability, and rigor. Transparent and reliable details of the search strategies and inclusion criteria will make the study even more transparent and reliable. Added methods for assessing and documenting biases will make the credibility of the review process even more robust.

2.3. Introduction of the review outcome

This literature review provides a possibility of a critical examination of available material related to the "Dimple-Augmented Ahmed Body Side Design" and its interaction with boundary layer turbulence in drag reduction. The objective is enhanced understanding through scrutiny of valuable contributions and their demarcation of shortcomings and inconsistencies within the discourse at the academic level. It is a process of critical literature review combined with case studies, an emphasis on establishing repeated patterns and weighing the pros and cons of the available methodologies in primary studies. Dimple augmentation seems quite imperative for improving aerodynamic efficiency in the Ahmed Body side design. Most researchers consider that dimples, mimicking those on a golf ball, would reduce turbulence in the boundary layer and drag. However, until now, all studies conducted on the issue have been inconclusive, and dimple augmentation efficiency has not been proven. Part of the research even casts doubt on the feasibility of using this method, while others claim it improves the flow separation patterns and vehicle economy.

It further points out that design modifications are closely related to flow dynamics and aerodynamic performance. Computational fluid dynamics simulations and experimental studies are two key approaches, although both methods do have some limitations. Inconsistencies occur with differences in the methods applied to the studies and environmental conditions. In the fusion of theoretical input and empirical findings, automotive aerodynamics can advance, facilitate innovation, and enhance vehicle efficiency and sustainability.

2.4. Optimizing aerodynamic efficiency through Rear-End modifications in Ahmed Body models

In automotive engineering, research into the reduction of aerodynamic drag is very crucial since it impacts both the fuel efficiency and performance of the vehicle. The Ahmed body has been developed as a simplified model for the study of car aerodynamic phenomena. This creates focus on some passive techniques of flow control, aiming at reducing drag on Ahmed body designs. (Mohammadikalakoo et al., 2020) researched passive drag reduction using linking tunnels at the rear of the Ahmed body. Linking tunnels allow for the passage of high-pressure air at sidewalls to the wake area, which is at low pressure, hence reducing wake size and its consequential pressure drag. The wind tunnel studies conducted showed a drag reduction of up to 5% using this approach.

(Siddiqui & Chaab, 2021), in their paper, proposed a rectangle flap mounted on the inclined surface of a 35° Ahmed body. A maximum drag reduction of 14% was observed by adjustment of the flap angle at 10°. This flap delays the primary separation point, prevents separation bubbles from forming, and reduces wake size, which in turn restores pressure and diminishes drag. (Koppa Shivanna et al., 2021) conducted an investigation on various rear cavity configurations over a square-back Ahmed body. Among these, they established that a tapered cavity gave the largest reduction in drag, which came out to be 22.55%. This reduction was attributed to a drop in wake size and an enhanced pressure distribution. (Zhai et al., 2023) investigated in their paper how different wheel spoke architectures would modify the aerodynamic efficiency of a 35° Ahmed body. By design parameter optimization of the wheel spokes, they could reduce drag significantly by a maximum of 4.7%.

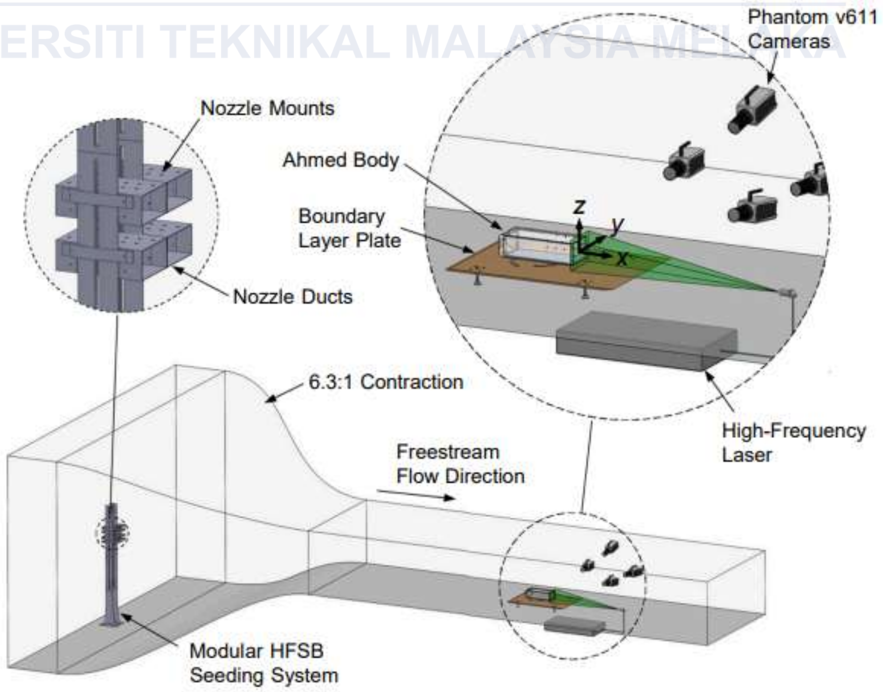


Figure 2.4.1: A schematic of the experimental setup showing the wind tunnel, Ahmed body, flat-plate, 3D-PTV system, and the nozzles used for generating the helium-filled soap bubbles. (Booysen et al., 2022)

Advanced numerical and experimental techniques have helped in going deep into the flow dynamics around Ahmed bodies. In their study, (Podvin et al., 2020) applied Proper Orthogonal Decomposition to investigate the wake flow patterns at the square-back Ahmed body. They extracted the most energetic flow modes and proved the importance of three-dimensional analysis in understanding wake energy distribution. (Maulenkul et al., 2021) suggested in their work an arbitrary hybrid turbulence modelling technique to simulate unsteady turbulent flows. This combines many turbulence models chosen by local mesh refinement. The method provided was checked using an Ahmed body and proved to be more performance-effective than the rest of the models. In a previous work, (Booysen et al., 2022) used 3D PTV to investigate the wake behind a square-back Ahmed body at high Reynolds numbers and cross-flow conditions. Their results showed a variation of wake structure with yaw angles. In another work, (Liu et al., 2021) conducted wind tunnel tests using Auto-PIV to measure the wake behind the DrivAer model, an automobile model representative of real-life conditions. They found out that due to the smooth contours, the flow separation is avoided and the recirculation bubbles are much smaller, thus contributing towards smaller pressure drag. It supports the consistent aerodynamic properties of the model.

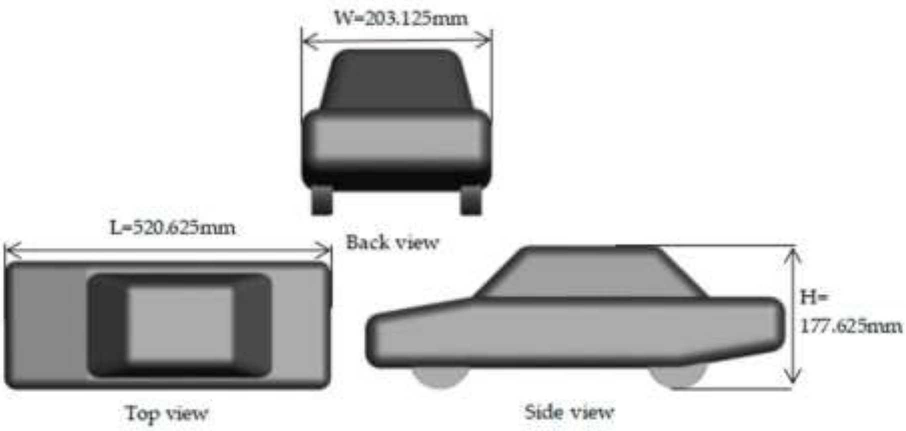


Figure 2.4.2: Motor Industry Research Association (MIRA) Geometry (Lai et al., 2020)

(Lai et al., 2020) also investigated the use of surface dielectric barrier discharge plasma actuators in active flow control over a notch-back automobile model. Plasma actuators installed at different locations near the rear changed the separation of the airflow and woke shape, which contributed to a substantial drag reduction of up to 13.17%. It is merely an example of what plasma actuators can do in car aerodynamics. In summary, this research in passive flow control methods using the Ahmed body has provided big drag reductions through a variety of diverse and creative means. This advanced our understanding of the flow dynamics and delivered crucial knowledge for improvements in aerodynamic optimization in the future.

2.5. A Benchmark model for aerodynamic drag reduction and flow control in vehicle design

The Ahmed body, as first proposed by S.R. Ahmed in his seminal publication in 1984 (Ahmed et al., n.d.), has since become one of the standard elements in vehicle aerodynamics, representing a model for studying complicated interaction between vehicle form and airflow. Ahmed's early work was focused on understanding the flow patterns around a bluff body with an inclined rear, thus giving out very important information about drag forces that impact vehicle performance. This model, as such, represented a basic geometric form with variable slant angles in such a way that would enable the researcher to systematically study the effect of different rear-end geometries on the aerodynamic drag.

Ahmed body work uncovered some of the most important phenomena contributing to drag, including flow separation, wake creation, and vortex shedding. His research showed that the angle of inclination at the back portion has a huge effect on the size and nature of the turbulent region behind it. The slant angle of 25° produced a well-defined separation bubble and attached wake, important features when drag forces are estimated. The Ahmed body, in his work, was developed as a benchmark reference model to test the accuracy of CFD simulations and experiments in vehicle aerodynamics.

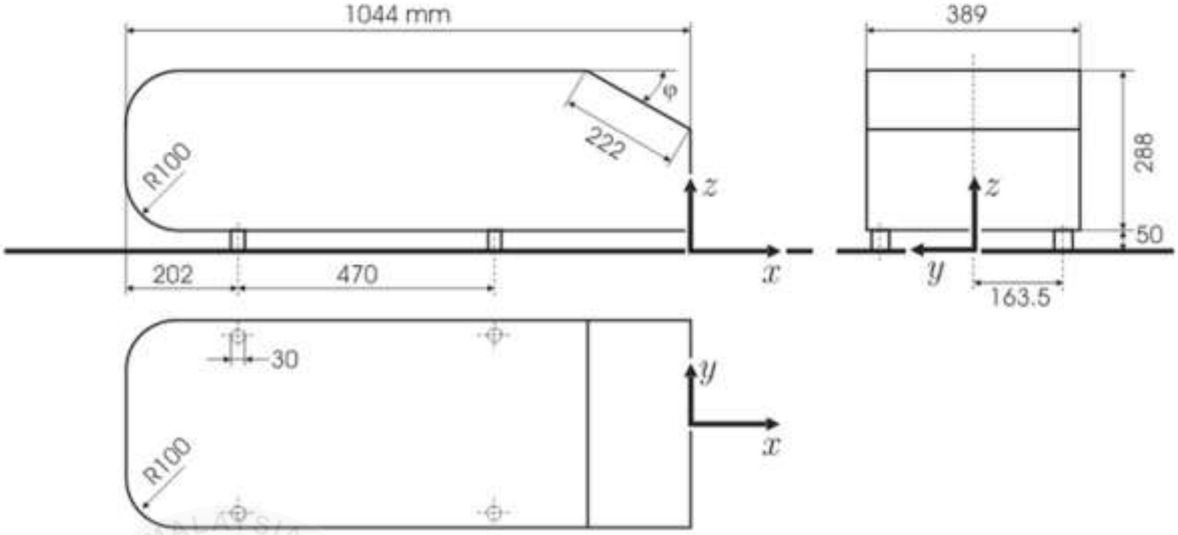


Figure 2.5.1: Ahmed body dimension (Mohammadikalakoo et al., 2020)

Further research used Ahmed's findings, and many passive and active flow control methods were tried with the aim of reducing drag for better fuel efficiency. For example, (Mohammadikalakoo et al., 2020) did some tests using rear linking tunnels that transferred high-pressure air from sidewalls into the wake area and experienced significant drag reduction. (Siddiqui & Chaab, 2021) presented a simple flap device, which efficiently controlled flow separation and enhanced pressure recovery in the study. Drag reduction showed a considerable contribution to experiments on the 35° Ahmed body.

Ahmed body work has influenced the development of advanced numerical methods. For example, (Maulenkul et al., 2021) applied a hybrid turbulence modelling methodology within Open-FOAM, using the Ahmed body as a case for simulation validation. Such methodology brings together RANS, URANS, LES, and DNS models with an attempt to combine high accuracy and low cost in simulations of highly complex flow phenomena.

Other successive studies have investigated the effect of different types of alterations made to the Ahmed body on its aerodynamic performance. (Koppa Shivanna et al., 2021) studied a set of designs for the rear cavity and observed that a tapered cavity can effectively reduce drag, by changing the wake shape. (Zhai et al., 2023) conducted studies on the optimization of wheel spoke structures. They found that drag can be reduced with certain designs by affecting the boundary layer and wake flow.

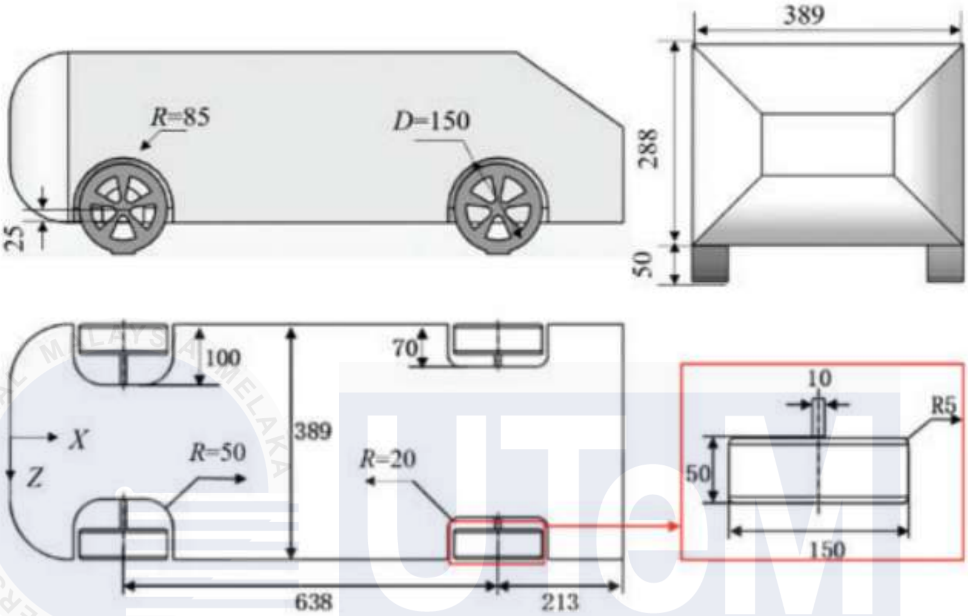


Figure 2.5.2: Ahmed body with four wheel (Zhai et al., 2023)

The Ahmed body has remained a fundamental but very crucial tool in research into vehicle aerodynamics and hence helps develop and check new methodologies that reduce drag. His work laid the foundation for further growth in the industry, firmly setting the framework through which the aerodynamic features of cars could be studied and gave substantial support for the production of more fuel-efficient and environmentally friendly automobiles.

2.6. Numerical and experimental approaches to enhancing aerodynamic efficiency in automotive design

Experiments on aerodynamics have involved new ways of analysis and optimizing flow properties around the model of the vehicle, with a focus on lowering drag. Computational Fluid Dynamics is a famous technique for gaining intricate understanding about performance in aerodynamics across different situations. In 2021, (Yudianto et al., 2021) conducted a computational fluid dynamics study of the aerodynamic coefficients of bus platooning in crosswinds. They have also proposed meshing algorithms that captured yaw angle and inter-bus distance effects correctly. To this end, (Siddiqui & Chaab, 2021) utilized the FLUENT software to investigate the effects of changing the flap angle of a rectangular flap overflow behaviour and separation bubbles. Their work was focused on studying a simple passive device for drag reduction on Ahmed body.

Of equal importance are experimental techniques. In this regard, (Liu et al., 2021) presented an automated particle imaging velocimetry method called Auto-PIV for studying the three-dimensional wake structure of the DrivAer model. It provided high accuracy in measuring the time-averaged velocity field distribution and the spatial distribution of dominant coherent structures. In this paper, (Maulenkul et al., 2021) presented an arbitrary hybrid turbulence modelling method, AHTM, in Open-FOAM. That method brought together a variety of turbulence models to simulate unsteady, flow-separated, turbulent flows that improved the accuracy and flexibility of aerodynamic design optimization.

Extensive studies were also conducted on passive flow control technologies. (Mohammadikalakoo et al., 2020) worked on the interaction of the car's body with the wind flowing around the body to minimize drag. In their study, the installation of connecting tunnels at the back of a bluff body implemented by researchers to redirect the high-pressure flow towards the wake area, mitigating pressure drag. It has numerically and experimentally attained a maximum drag reduction of 5%. (Podvin et al., 2020) applied the POD-based analysis and simulation to understand the flow dynamics of the square back of Ahmed body. They were successful in capturing all the main features, like anti-symmetrical quasi-steady deviation modes and mechanisms related to vortex shedding.

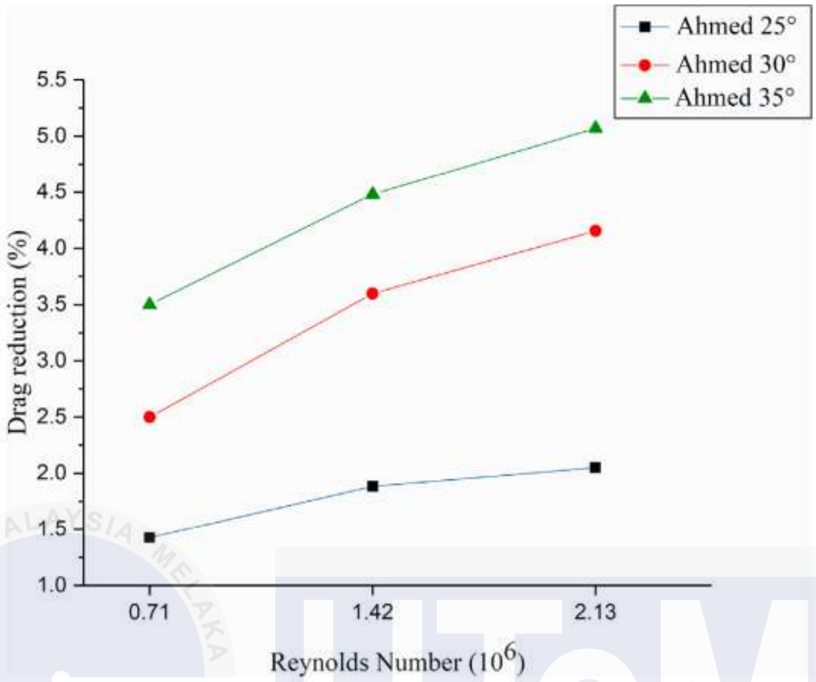


Figure 2.6: Drag reduction of passive flow control for different slant angles.

(Mohammadikalakoo et al., 2020)

Active flow control techniques also offer some promises. In their paper, (Lai et al., 2020) investigated the possibility of using surface dielectric barrier discharge plasma actuators in controlling the airflow separation at the rear of notch-back vehicles. Numerical simulations on phenomenological grounds by these authors showed a substantial reduction in drag due to plasma actuation applied at multiple spots simultaneously. The drag reduction coefficient obtained is maximum at 13.17%. In the paper by (Yagmur et al., 2020), the authors used RANS, DES, and LES turbulence models to get the flow characteristics around a semi-circular cylinder. The researchers found that the LES and DES models were more reliable compared to the RANS models since it showed better agreement with the experimental data.

(Verma et al., 2020) in their study considered the problem of turbulence drag reduction using magneto-hydrodynamic turbulence. They found that in comparison to hydrodynamic turbulence, MHD turbulence exhibits reduced kinetic energy flow and turbulent drag. In their efforts, (Zhang et al., 2022) applied large-eddy simulation to examine the role of driblet drag reduction on an infinite swept wing under low Reynolds number conditions. According to them, the maximum drag reduction ratio was 9.5%. (J. Wang et al., 2020) analysed the impact of bogie cavities and bogies on high-speed train aerodynamic performance; in the process, they applied a highly advanced simulation technique, called IDDES, and noticed that there were considerable decreases in turbulent kinetic energy and drag with either sealing the cavities or including bogies.

In the paper of (Yu et al., 2020), the wake turbulence behind a circular cylinder has been considered using advanced spectral/hp methods for manipulation by use of counter-rotating rods. The authors have managed to reduce drag by 25.1% by stabilizing the wake behind the cylinder. (McDermott et al., 2020)enhanced a viscoelastic turbulence model to better forecast drag reduction in channel flow. This development enables the proper modelling of turbulent kinetic energy at any point in the channel and is quite independent of friction velocity. (Koppa Shivanna et al., 2021) conduct numerical simulations over the SBAB with different cavity modifications at the rear. They found that tapered cavities were responsible for the maximum drag reduction due to the shift in the wake's average structure.

(Cafiero et al., 2024) applied particle image velocimetry to quantify the drag reduction imparted by sinusoidal driblets. Small grooves perturb the near-wall arrangement of the boundary layer, causing a substantial reduction in turbulence generation and eventually friction drag. (Ricco et al., 2021) reviewed methods for reducing turbulent skin-friction drag. They indicated that the near-wall space has effective transverse force control of boundary layer turbulence and drag reduction.

It was noted that multiple methodologies underline the necessity of a wholesome strategy to understand and improve the aerodynamic characteristics of vehicle models for the reduction of drag, through the coupling of numerical simulations with experimental techniques.

2.7. Advancements in CFD Techniques for Drag Reduction in Vehicle Aerodynamics

Nowadays, CFD is one of the very important fields in automobile engineering. Among all its usages, the most prominent objective is the improvement of fuel efficiency and performance by the reduction in drag of vehicles.

Table 2.7: CFD Investigation in the previous

Author	Tools	Turbulence Model	Flowrate	Remarks
(Mohammadikalakoo et al., 2020)	ANSYS Fluent	k-ω (SST) model k-ε model	10, 20, and 30 m/s	Passive flow control via rear linking tunnels
(Lai et al., 2020)	STAR CCM+	k-ω (SST) model	30 m/s	Aerodynamic Drag Reduction and Optimization of MIRA Model
(Maulenkul et al., 2021)	OpenFOAM/ DAFoam	k-ω (SST) model k-ε model URANS VLES DES-SST LES-SVV	40 m/s	Efficient and Accurate Automotive Aerodynamic Analysis and Design Optimization
(Siddiqui & Chaab, 2021)	ANSYS Fluent	k-ω (SST) model k-ε model	40 m/s	Passive rectangular flap device on the 35° Ahmed body
(Koppa Shivanna et al., 2021)	Open-FOAM	k-ω (SST) model	50 m/s	Impact of various passive flow controller designs on drag reduction and mean wake topology in turbulent flow over a Square Back Ahmed Body (SBAB)
(Zhai et al., 2023)	ANSYS Fluent	k-ω (SST) model	40 m/s	Optimization of Wheel Spoke Structure for Drag Reduction of an Ahmed Body
(J. Wang et al., 2020)	STAR CCM+	k-ω (SST) model	13.8 m/s	Effects of bogie cavities and bogies on the aerodynamic behavior of a high-speed train (HST)
(Yudianto et al., 2021)	ANSYS Fluent	k-ω (SST) model k-ε model	10 m/s	Aerodynamics of Bus Platooning under Crosswind
(Bonnavion et al., 2022)	ANSYS Fluent	k-ε model	39 m/s	Kinetic energy balance for the volumetric identification of drag sources of a blunt body. Application to road vehicles
(Mondal et al., 2023)	ANSYS Fluent	k-ω (SST) model k-ε model	19.245 m/s	Investigation of on-demand fluidic winglet aerodynamic performance and turbulent characterization
(Yudianto et al., 2022)	ANSYS Fluent	k-ω (SST) model	1.76 m/s, 3.64 m/s, and 5.77 m/s.	Aerodynamic Characteristics of Overtaking Bus under Crosswind
(G. Chen et al., 2022)	STAR-CCM+	k-ω (SST) model	60 m/s	Effect of train length on train aerodynamic performance

Passive and active flow control techniques are under practice in the research done in the recent past for huge drag reduction. (Mohammadikalakoo et al., 2020) investigated the role that the rear linking tunnels play in passive flow control over an Ahmed body. The study taken up by these researchers was a quantitative one with empirical analysis, and according to their claims, this technique has been able to reduce drag successfully by approximately 5%. This high pressure at the sidewalls is concentrated in flow and effectively reduces the amount of wake and, accordingly, the pressure drag. Similarly, (Siddiqui & Chaab, 2021) proposed another simple mechanism for a rectangular flap that can be used in reducing drag on a 35° Ahmed body. Their computational fluid dynamic study through ANSYS Fluent showed a drag reduction of 14% by changing the flow conditions at the rear of the model, which prevents the separation bubble from forming inside the slant volume. (Koppa Shivanna et al., 2021) have conducted a study on the effects of the rear cavity modification on Square Back Ahmed Body using Open-FOAM. These researchers, however, reported that drag was significantly reduced by 22.55% owing to the tapering of edges. This reduction was attributed to a decrease in the size of the wake and an improvement in base pressure distribution.

Active control of flow techniques, on the other hand, are quite promising. In the work of (Lai et al., 2020), an experimental investigation into the application of SDBD plasma actuators to control airflow separation on notch-back automobiles was conducted. Their study showed that plasma actuation at different points reduced the drag coefficient up to 13.17%. The most encouraging result was acquired when the actuation was at four locations simultaneously. In the study by (Mondal et al., 2023), fluidic winglets on an aircraft with low aspect ratio wings were investigated. The main claim of these researchers is that winglets generating jets that counteract the tip vortex, as proposed, turn out to be efficient in drag reduction without actually increasing the weight of the structure. This effect was most pronounced during low-speed flight conditions.

There have been huge developments in methods for turbulence modelling. (Maulenkul et al., 2021) presented a new advanced hybrid turbulence modelling method (AHTM), where RANS, URANS, LES, and DNS models are unified in one single flow field. Their solution, based on Open-FOAM, has proven to be good in terms of accuracy and flexibility when applied to the optimization of vehicle aerodynamic designs, more especially in more complex turbulent phenomena, such as Kelvin-Helmholtz instability.

Research into high-speed train aerodynamics has unveiled a number of important findings. (J. Wang et al., 2020) carried out a detailed investigation on the effects of bogies and cavities on the aerodynamics of a HST using an improved IDDES. Their simulations results indicated huge reductions in turbulent kinetic energies and slipstream velocity on specified configurations, amounting to drag reduction by 56.9% when the cavities were closed. Chen et al. did an analysis on the effect of train length through the IDDES in their work. The results showed that with the increase in the size of the train, there was an increase in the thickness of the boundary layer and turbulence kinetic energy. It might be a result of increased drag and lift coefficients. By comparing the result with experimental data, the numerical model was checked to be reliable in estimating aerodynamic performance.

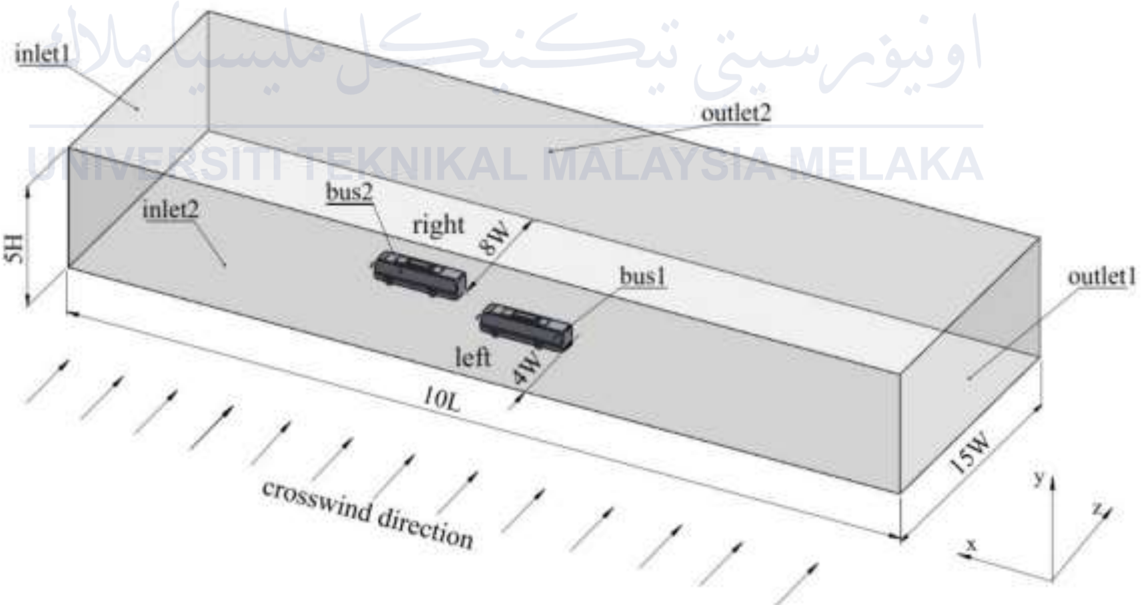


Figure 2.7.1: Computational domain of the overtaking bus (Yudianto et al., 2022)

Also praiseworthy is the research being conducted on bus aerodynamics under cross wind conditions. (Yudianto et al., 2021, 2022) employed ANSYS Fluent to investigate the aerodynamic characteristics of overtaking and platooning under several crosswind situations.

It has been demonstrated that the results had a significantly negative impact on the aerodynamic advantage produced by platooning. Two basic characteristics of flows affected the buses: the distance between them and their yaw angle in respect to the wind. These parameters do heavily influence flow patterns and aerodynamic coefficients.

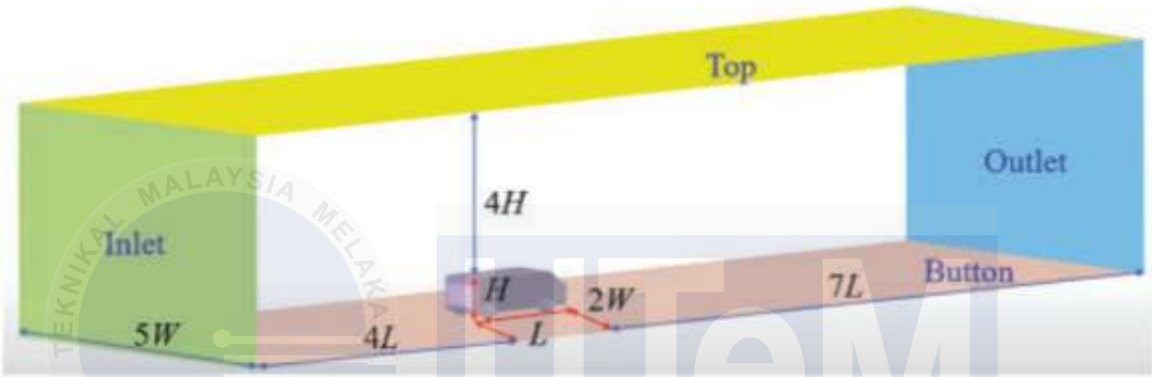


Figure 2.7.2: Computing domain and boundary conditions (Zhai et al., 2023)

Most research in this scenario is under process to improve the efficiency of aerodynamics. (Zhai et al., 2023) studied ANSYS Fluent for the problem of drag reduction through the optimization of the wheel spoke structure over an Ahmed body. They used a Kriging model and thereafter an adaptive simulated annealing algorithm to find the best design variables. Accordingly, it was shown that they could achieve maximum drag reduction on the Ahmed body to the tune of 4.7%. (Bonnavion et al., 2022) presented a method to identify drag sources in vehicle aerodynamics based on the principle of kinetic energy balance. Their concept of a multi-turbulence model applicable methodology made it possible to perform an accurate optimization of Drag Reduction Techniques through volumetric loss breakdown in distinct flow areas.

The literature review shows that over the years, there has been tremendous growth in CFD methods in reducing vehicle drag. These include passive and active flow control methods and new turbulence modelling approaches for applications of these methods to various vehicle classes. There is ongoing research in this area that might result in further improvements in vehicles' aerodynamics and hence fuel efficiency and performance.

2.8. Advancements in turbulent kinetic energy analysis for vehicle aerodynamics

On the aerodynamic performance, mostly relevant to vehicle models like the Ahmed body, TKE has undergone exhaustive development in the recent past with the help of various computational and experimental methodologies. In an all-encompassing study, (Yagmur et al., 2020) studied the flow over a Semi-Circular Cylinder (SCC) using various models for turbulence: Reynolds-Averaged Navier-Stokes, Detached Eddy Simulation, and Large Eddy Simulation. Their study showed that with LES and DES, results closer to the experimental data were obtained, with deviations in the drag coefficient less than 7%, thus showing an inclination towards more accurate TKE analysis in unsteady flow structures. (Verma et al., 2020) did a study on magneto-hydrodynamic turbulence dynamics in comparison to hydrodynamic turbulence. Their results showed that MHD turbulence diminishes kinetic energy flux and nonlinear terms in the equation, which in turn further reduces turbulent drag. This theoretical prediction has also been supported by their shell model simulations, proving very large possibilities of MHD turbulence in reducing drag.

(Yu et al., 2020) did a DNS study on the effect of small counter-rotating rods on wake turbulence in the background of a circular cylinder. Their results depicted a current drag reduction of around 25.1% during high Reynolds numbers because of the pressure recovery effects generated by the spinning rods. The application of the method was quite successful in stabilizing the wake and also in reducing TKE in turbulent regimes. (Zhang et al., 2022) performed a comparative study of triangular driblets with respect to their effects on compressive flat-plate flow turbulent boundary layers. Using DNS, they revealed that the driblets caused global drag reduction by both increasing the logarithmic-law region of the boundary layer and reducing the wall friction velocity, Reynolds normal stresses and TKE production. They underlined in their work the two roles of the driblets in controlling the drag: lifting and rectification. (Lloyd et al., 2023) performed DNS and RANS model simulations of flow over shark skin denticles. Their results indicated that denticles increase drag by 58% relative to a flat plate due to high-speed fluid penetration between the denticles. Although RANS models under-estimated the production of TKE, they were found to be capable of predicting drag distributions, thus remain effective tools for understanding mechanisms of drag.

(X. Chen et al., 2024) utilized phase-locked stereo PIV to investigate the effect of sweeping jets on after-body vortices shed from a slanted base cylinder. Their results indicated that the sweeping jets increase TKE in the vortex region, hence reducing the velocity gradient and increasing turbulence ingestion. Hence, this method showed promise for application soon in after-body vortex control and performance improvement. (Serafini et al., 2024) performed DNS of turbulent pipe flow with dilute polymer solutions to investigate how mean and TKE budgets are modified. Their study showed that polymers reduce the Reynolds stress and turbulent production, acting as a source of TKE at higher Weissenberg numbers. Such effects, more important for low Reynolds numbers, showed that polymers play an important role in the manipulation of TKE distributions and the establishment of drag reduction.

(Cafiero et al., 2024) examined the effects of sinusoidal driblets on turbulent boundary layers. Their experimental results demonstrated that sinusoidal driblets showed considerable suppression of turbulence production with the fragmentation of low-speed streaks and the weakening of stream-wise vortices. This kind of modulation of the turbulent regeneration cycle provided significant drag reduction and thereby proved the efficacy of sinusoidal driblets for aerodynamic applications. (Gattere et al., 2024) investigated the impact of stream-wise-traveling waves on turbulent skin-friction drag in compressible regimes in 2024. Their DNS results indicated that compressibility increases drag reduction benefits at lower frequencies and wavenumbers. It was suggested that by controlling the bulk temperature to match the aerodynamic heating, one has more realistic test conditions for making a marginal improvement in drag reduction under compressible flow conditions.

It brings out sufficient improvement in the methods of TKE investigation to enhance aerodynamic performance. Research with sophisticated models of turbulence, innovation in flow control strategies, and comprehensive analyses of the aerodynamic behaviour bring out valuable insight into TKE distribution and drag reduction techniques. Much research in such areas is underway and holds promise for much improvement in vehicle aerodynamics and efficiency.

2.9. Advancements in turbulence modeling and flow control for aerodynamic efficiency

This is an in-depth review of methodologies and research toward the understanding and improvement of aerodynamic efficiency. This paper synthesizes the findings of different methods and summarizes major developments in the area of study.

(Fowler et al., 2020) conducted scale-resolving simulations using the PANS k-ω closure model to find out the turbulent wake of a square cylinder at a Reynolds number of 22,000. They used different levels of resolution to infer the required precision for several flow characteristics, including flow statistics and coherent structures. In the present work, the authors have introduced two measures that especially target coherent structures: Fourier and Chebyshev decompositions. Results showed that larger resolutions are required to accurately predict both integral values and coherent structures. (Kamble et al., 2020) proposed a two-layer PANS model for improving the wall-bounded simulations' accuracy. The PANS k–ε turbulence model is combined for the outer region, and an unresolved kinetic energy equation in the inner layer is modelled. In the current study, it has been shown that the two-layer PANS model can accurately simulate channel flow; thus, it can be rated as a pretty promising tool for performing scale-resolving simulations. (Kamble & Girimaji, 2020) investigated the turbulent wake behind a sphere using Partially-Averaged Navier-Stokes simulations at different resolution levels. Accurate capture of coherent structures is contingent upon the sufficient resolution of the essential instabilities. Their highest-resolution simulation captured all large-scale organized structures, reproducing experimental data.

(Yousefi et al., 2020) used interface-resolved simulations to examine the modulation of statistically steady-state homogeneous shear turbulence by finite-size particles. A non-monotonic behaviour in the turbulent kinetic energy was observed as a function of increasing solid volume fraction due to interactions between the particles. A key focus of the study was contrasting the behaviours of spherical and oblate particles. The results returned that the rotational rates in oblate particles were much higher, and this affects the modulation of turbulence to a far greater degree than in spheres. (Yang et al., 2021) made direct numerical simulations of particle-laden downward channel flows, investigating how the terminal velocities of particles affect the modulation of turbulence. It was observed that the presence of heavy particles increased the average velocity of fluid at the centre of the channel. On the contrary, in the buffer layer, heavy particles decreased the average velocity. More importantly, it accumulated along walls and significantly disturbed the structures of turbulence.

(Li et al., 2021) conducted a study on active flow control using twin synthetic jets around a wake behind a finite-length square cylinder. The results from their study showed that the dual synthetic jets had an intensive effect on the reduction of mean drag and fluctuating drag and lift coefficients. This was through variation in the momentum of the jets and their frequency of drive. It was represented by the flow visualization that there was considerable suppression of the separated shear flow, and increased turbulent kinetic energy towards the free end of the cylinder. (Lin et al., 2022) conducted a study on the interaction between polymers and turbulence in duct flow with major consideration of drag reduction. In the study, it was presented that the introduction of polymers lowered effectively the turbulent kinetic energy and modified the Reynolds stress tensor, which finally reduced drag by improving the dissipation of turbulent fluctuations.

(Hamada et al., 2023) investigate, using direct numerical simulations, the role of distributed roughness in reducing drag during transitional flow states. Their work has shown that certain roughness configurations exert powerful drag reduction by altering the properties of the boundary layer and turbulence structures. (L. Wang et al., 2022), the performance of a compressor cascade with high load dependent upon the ratio of dimple depth to its diameter. Three different flow regimes were identified inside the dimples, which influenced energy distribution and turbulence characteristics, finally affected the cascade performance. Results from their study showed the optimization of dimple configurations is able to reduce corner separation effectively. (Rao & Liu, 2020) used large-eddy simulations to conduct research on the effect of the Reynolds number upon the aerodynamic characteristics of the leading-edge serrations bio-inspired from owls. It was found out that these tiny notches at the leading edge efficiently managed this change from smooth to turbulent air flow and enhanced the lift on to drag ratio at higher Reynolds number. Hence, which means these could be employed for noise reduction in wind turbines and performance enhancement in multi copters.

(McDermott et al., 2021) have been developing an improved k-ω turbulence model in order to forecast complex turbulent flows including the polymeric solutions. Their model performed quite well in several regimes of drag reduction, offering a much simpler and more efficient way of modelling viscoelastic fluid flows than previous attempts—verified against direct numerical simulation data. The works reviewed above show considerable developments in understanding and controlling turbulent flows.

Practical applications of the findings range from flow control around bodies to drag reduction in various aerodynamic configurations. These results give an indication that the overall objective of improving aerodynamic performance by using advanced simulation techniques and experimental testing can be met.

2.10. Summary

The research into applying dimple-augmented Ahmed body designs for drag reduction and turbulence control has returned some marvellous findings and brought forward particular areas that could be improved. Current models of turbulence, like the RANS and DES approach, often under predict the generation of TKE, hence predicting the drag with inaccurate results. Added to this has been the rapidity of penetration of fluid through structures like denticles, thereby making the need for more accurate modelling techniques all the more vital.

Present interest in the aerodynamics research, there has been recent interest in using advanced computational fluid dynamics techniques coupled with state-of-the-art flow control technologies aimed at reducing drag.

Research has proven passive and active flow control methods, such as sinusoidal driblets, sweeping jets, and polymers, all work on the manipulation of turbulence for drag reduction. State-of-the-art imaging techniques, such as phase-locked stereo PIV, deliver exquisite insights into aerodynamic behaviour and have already helped to improve the performance of vehicles. Improvements in the future will be realized through improved models of turbulence that better capture TKE dynamics and drag reduction strategies, as well as advanced flow control methodologies like magneto-hydrodynamic turbulence and synthetic jets. In this manner, it will be possible to get an in-depth understanding of vehicle aerodynamics, hence developing more efficient ways of drag reduction through combined numerical simulations and experimental approaches. To put it otherwise, much as what has so far been realized is to be taken on board, equal interest and effort have to be channelled toward the limits existing in modelling and exploring emerging trends and innovations that look toward the future on vehicle aerodynamics.

CHAPTER 3

METHODOLOGY

3.1. Introduction

This chapter further elaborates on the methods adopted to study the effects of dimple augmentation on the side surfaces of the Ahmed body. The major focus of this study is to analyse the turbulence kinetic energy and drag reduction for this configuration. The methodology incorporates a combination of computational fluid dynamics simulations and experimental validation, which allows in-depth understanding of the aerodynamic impacts of the dimple modifications. The use of different methodologies ensures a critical investigation using knowledge both at the conceptual and applied levels to address the gaps that exist in the literature.

The Ahmed body comprises a simplified representation of the vehicle geometries and is a basic model for the study of the flow phenomena around a body vehicle, especially in terms of drag and wake production (Ahmed et al., n.d.) Previous works have established the fact that changes in the surface structure, especially by using dimples, can cause a significant effect in the flow dynamics around the vehicle, with possible drag reductions (Koppa Shivanna et al., 2021; Viswanathan, 2021). However, the exact mechanisms of how dimples affect turbulent kinetic energy (TKE) and drag with a reference to the Ahmed body have not been thoroughly investigated. As a result, the task of this research is to close this gap by conducting in-depth CFD calculations and controlled experimental setups.

CFD analyses are employed as an effective tool to visualize and evaluate the flow patterns around modified vehicle designs. The present task will involve advanced simulation techniques to evaluate the impact of dimple augmentation on flow separation, reattachment sites, and wake dynamics. The effects of dimple augmentation have been analyzed (Lehmkuhl et al., 2019; Mohammadikalakoo et al., 2020). Wind tunnel experiments will be linked to these simulations, which will provide empirical results to validate computational models, checking if the models are precise enough to predict the actual aerodynamic behavior (Akhter et al., 2022; Luo et al., 2022). By using this dual methodology, one can attempt a proper evaluation of the changes one can see in the aerodynamic performance with dimple augmentation in place.

In other words, the methodology section will detail the methodologies and techniques used for the analysis of the aerodynamic consequences of dimples in the Ahmed body. This paper combines CFD simulations with experimental validation to conduct a fully reliable and comprehensive analysis of drag reduction and TKE with a view to dimple impact in automotive aerodynamics. The implications of the results are discussed.

3.2. Equipment

i. Catia:

Figure 3.2.1: Catia V5 R21

Used for Computer-Aided Design (CAD) to create the initial geometry of the Ahmed body and wind tunnel. Catia offers advanced sketching and modeling features, enabling precise construction of complex aerodynamic shapes necessary for realistic simulation. Its robust design capabilities allow for detailed modifications, such as dimple augmentations and boundary structures, which are critical to evaluating turbulence and drag effects.

ii. Ansys Workbench:

Figure 3.2.2: Ansys Workbench

Used for meshing and simulation setup. Ansys Workbench provides an integrated platform for importing and refining CAD models, setting up meshing parameters, and configuring boundary conditions essential for accurate CFD analysis. The flexibility in Ansys Workbench’s meshing tools, such as automatic inflation and body sizing, allows fine control over element size and distribution, especially in high-turbulence zones.

iii. Ansys Fluent:

Utilized for Computational Fluid Dynamics (CFD) simulations. Ansys Fluent handles the numerical solution of fluid flow and heat transfer equations, essential for assessing drag and turbulence kinetic energy in aerodynamic studies. With its advanced solver options, Fluent facilitates the simulation of airflow around complex shapes, including the Ahmed body with dimple modifications. Fluent's robust turbulence models, including Reynolds-averaged Navier-Stokes (RANS) equations, provide detailed insight into boundary layer behaviors and wake formations.

These software tools, when used together, enable a comprehensive workflow from design and meshing to simulation, ensuring accurate and reliable results in the study of aerodynamic drag reduction techniques.

3.2.1. Dassault Systems: Catia V5 Software

CATIA V5, designed by Dassault Systems, is a leading computer-aided design (CAD) program broadly used in industries such as the automotive and aerospace industries. CATIA V5 is considered quite strong in accurate geometric modeling and sophisticated surface design. It is, in this respect, specifically useful for designing the Ahmed Body, a widely used model in aerodynamic research. The parametric modeling features of the software make it easy to modify dimensions and other features such as the slant angle and length. In this way, it allows for the study of various configurations and their influence on aerodynamic performance.

The GSD module of the software is very efficient in creating smooth and accurate surfaces necessary to optimize aerodynamic performance. CATIA V5 also supports feature-based design that makes it easy to add and modify dimples on the surface of the Ahmed Body. This flexibility allows designers to systematically evaluate many patterns of dimples and optimally achieve drag reduction and control of boundary layer turbulence. It can be easily integrated into CAE tools like ANSYS Fluent in order to export high-quality geometries for further aerodynamic analysis.

Furthermore, CATIA V5 comes with high-level visualization and technical documentation tools that help to fully understand and communicate design changes. The assembly design of the Ahmed Body enables detailed studies involving its assembly with other vehicle body components. CATIA V5 is an important tool for designing and optimizing the Dimple Augmented Ahmed Body; hence, it enables comprehensive research on drag reduction and control of turbulence.

3.2.2. ANSYS

ANSYS is one of the most important tools used in engineering simulations and provides a wide variety of functionalities, which make it indispensable in the analysis of complex physical processes. The Dimple Augmented Ahmed Body study leverages ANSYS Fluent, one of the most respected software in the market, offering excellent capabilities in the area of Computational Fluid Dynamics, or CFD. Employing Fluent, the engineers are enabled to conduct studies on the sophisticated turbulent flows around the Ahmed Body in order to realize the subtle effects of the dimple designs on the reduction of drag and the dynamics of the boundary layer.

In addition to its competence in CFD, ANSYS's tool suite now includes ANSYS Mechanical and ANSYS Thermal, for structural and thermal analysis, respectively. This holistic approach ensures a detailed investigation into the performance of the design across various situations, including structural soundness against a variety of loads and the influence of thermal effects on aerodynamic efficiency. ANSYS allows multiple skills to be seamlessly integrated together so that aero engineers design such aerodynamic solutions that maximize performance, are robust, and adaptable to real-world conditions.

3.3. Benchmark Process

The benchmarking process is fundamental to the scientific credibility of CFD results. It takes the data from computational results, away from theoretical forms and closer to real, verifiable information that can be applied with confidence to the design and analysis of engineering. The cycle of benchmarking, which consists of hundreds of numbers of validation and improvement cycles, is consistent with the technique of scientific investigations and the approach to the process of continuous development.

From an academic point of view, the value of benchmarking can be overemphasized. The systematic framework which is provided by this method offers better chances for detecting and eliminating mistakes. It enhances the robustness of numerical models and enables the user to put their trust into the predictive ability of CFD more firmly. Further, the documentation and sharing of benchmarking studies is a way that benefits the broader scientific community as it establishes proven methods and benchmarks, which can be used by others for comparison and further study as depicted in figure 4 below.

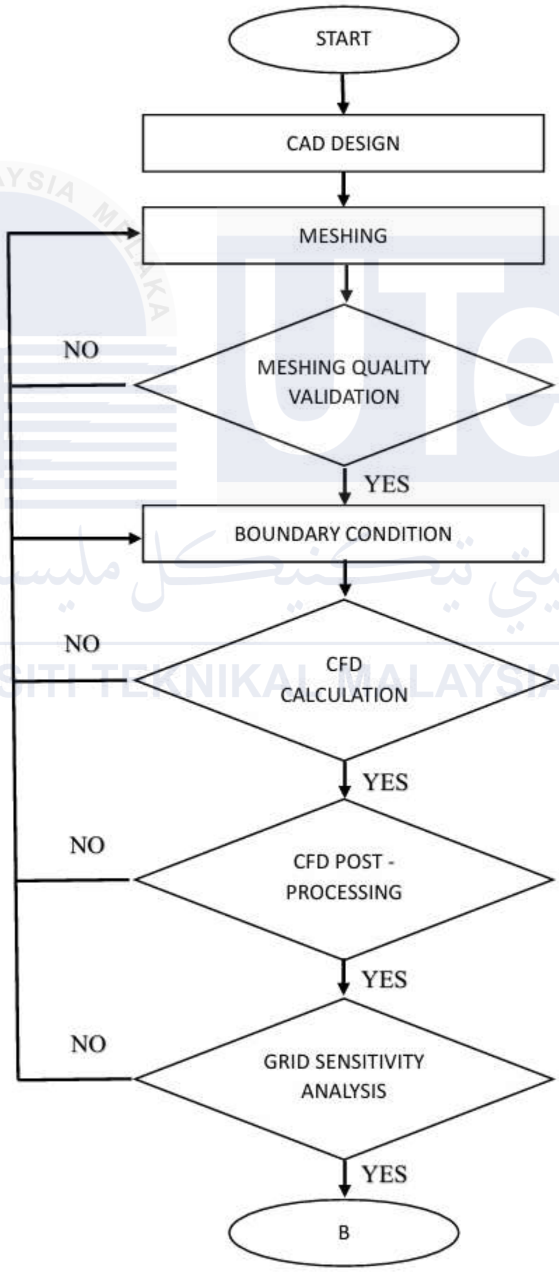


Figure 3.3: Benchmark Flowchart

The benchmarking procedure of CFD as shown in figure 4 is a rigorous and very important process in ensuring the validity and accuracy of simulation results. This is a systematic process involving careful selection of benchmark cases, rigorous quality review, comprehensive validation, clear formulation of boundary conditions, detailed numeric solution, thorough post-processing, and finally, a vigorous grid sensitivity study. CFD models can achieve high levels of accuracy and dependability during this process, making them important tools in both academic studies and real engineering applications. By applying this systematic method, experts can provide reliable simulations, offering immense knowledge to the field of fluid dynamics.

3.3.1. CAD Design

CAD modeling in aerodynamic simulations has to be as accurate as possible to represent real geometries and ensure that all the essential physical phenomena, like boundary layer formation, vortex shedding, and separation of flow, are faithfully captured. Construction of the Ahmed body was done using Catia software inside a standardized tunnel to create a controlled environment for systematic comparisons of turbulence characteristics and drag. The tunnel dimensions are chosen with due care for a sufficiently large domain in order to avoid any artificial boundary effects. These might include inaccuracies in the results of the simulation because of unnatural constraints on the development of the airflow.

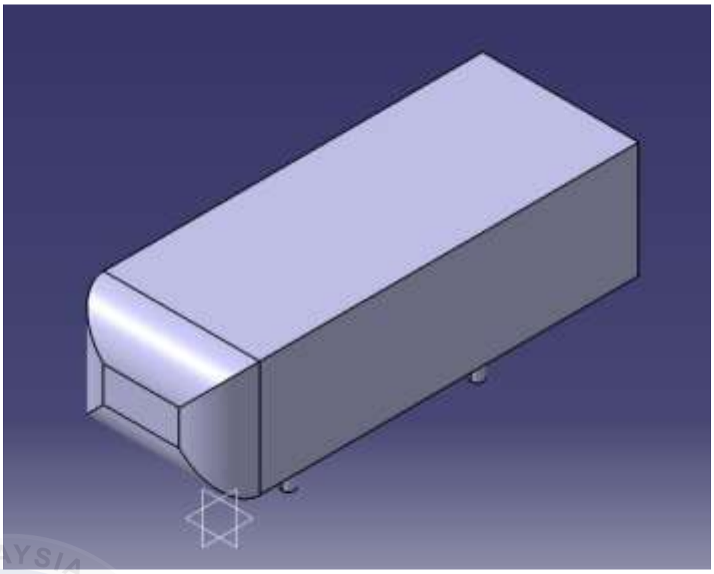


Figure 3.3.1: Ahmed Body Design

To analyse turbulence kinetic energy and assess drag reduction for the dimple-augmented side of Ahmed body, a detailed CAD model was created using Catia software as shown in Figure 3.3.1. The following design parameters and processes were followed:

a. Tunnel and Ahmed Body Design:

- A wind tunnel was sketched with dimensions: Length of 10,000 mm and Height of 4,000 mm.
- A symmetrical pad was generated with a Width of 2,000 mm.
- The Ahmed body was modelled within this tunnel according to standard Ahmed body dimensions to represent an automotive structure for aerodynamic analysis.

b. Wind Tunnel and Ahmed Body Components

Specific design enhancements were made to the Ahmed body and tunnel to simulate various drag-reducing features:

i. Carbox Design:

A carbox was modelled on the Ahmed body by sketching a rectangle, extruding it to some depth, and renaming as "carbox." In this carbox, it is used as the influence region for meshing and is done with an element size of 15 mm to capture flow separation points around this component effectively.

ii. Underbody Design:

An underbody box was similarly added and assigned an element size of 10 mm for detailed meshing to capture flow effects underneath the vehicle.

iii. Wake Box Design:

A wake box was sketched and extruded on the Ahmed body, representing the region where flow separation and wake formation occur. This box was meshed with an element size of 10 mm to ensure precise data on flow detachment and reattachment.

3.3.2. Meshing

The meshing in CFD acts to define the resolution with which the fluid flow will be solved within the computational domain. A well-constructed mesh can capture such subtle variations in velocity, pressure, and turbulent structures, especially in areas close to the surface where boundary layer effects are most pronounced. In the present work, the balance between geometric fidelity and computational efficiency has been struck by integrating the legs of the Ahmed body into a single face, in a selective freezing of the components. This will evade unnecessary expenses that over-complex geometries bring in but capture the essential features of the flow, important for the drag profile of the Ahmed body.

Inflation layers from the body surface are important for inflation in CFD analysis, since it provides modeling of a turbulent boundary layer. These inflation layers will be the bridge for high gradient regions near the wall to the free stream, enhancing the simulation in order to get better predictions for wall shear stresses. In this boundary layer-dominated field of automotive aerodynamics, even slight inaccuracies in the resolution of the boundary layer will result in considerable errors in the drag and lift predictions. An inflation layer, when properly configured with best practices from the literature, enables a realistic velocity profile along the vehicle's surface to be specified.

3.3.2.1. Wind Tunnel Enclosure Setup

The choice of a box-shaped enclosure with non-uniform cushions to simulate airflow around Ahmed body is based on a thorough understanding of airflow patterns typical of automotive environments. This design decision allows the model to maintain a realistic representation of the free stream conditions encountered by an actual vehicle as shown in Figure 3.3.2.1.

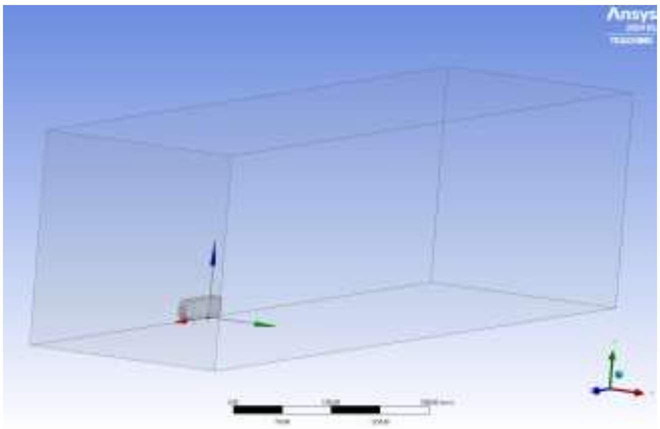


Figure 3.3.2.1: Enclosure on Ahmed Body

Additionally, using a Boolean subtraction method ensures a seamless air volume, enhancing the accuracy of airflow interactions around Ahmed body. This technique also mitigates artificial flow restrictions or numerical instabilities, which can distort results in high-turbulence regions, such as the wake and underbody.

3.3.2.2. Refinement and Optimization through Meshing

Enhanced meshing around critical areas, like the carbox and wakebox, provides increased resolution where turbulent effects are expected to be most intense as shown in Figure 3.3.2.2.

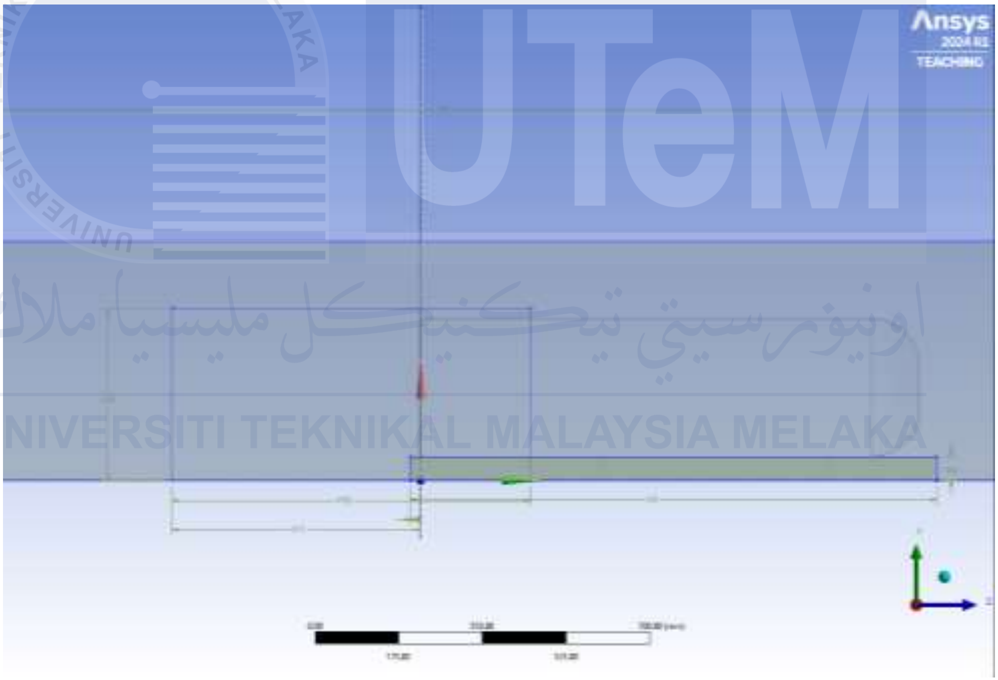


Figure 3.3.2.2: Carbox and Wakebox position

For instance, carbox meshing targets flow structures around the vehicle in formation and detachment to study in detail the drag effects induced by dimples. Similarly, wakebox design is instrumental for capturing the flow separation region at the very heart of the drag-force contributions. The academic studies are unanimous in the fact that a structured mesh with refinement in regions of high turbulence is superior in predicting complex flows compared to a uniformly coarse mesh, which may overlook the smaller eddies and vortices.

3.3.3 Meshing Quality Validation

Quality validation of the meshing is very important in ensuring that the CFD simulation will capture the complex behavior of aerodynamics around the body of Ahmed, mainly focusing on areas of turbulence kinetic energy and drag reduction. The validation started with a mesh independence study by refining the mesh density in highly turbulent areas, such as dimpled surfaces and the wake region. By incrementally increasing element density and observing convergence in key outputs such as drag coefficient and turbulence kinetic energy, the study verified that the mesh was sufficiently resolved, avoiding unnecessary computational costs while maintaining accuracy. Element quality metrics, including skewness and orthogonal quality, were optimized to reduce numerical errors, skewness was kept below 0.5 in critical flow regions, and orthogonal quality was maintained above 0.7 to ensure smooth alignment of elements with the flow direction, particularly around the Ahmed body and dimples.

Further refinement was targeted at boundary layer effects and areas where flow separation was likely to occur. Inflation layers for resolving the boundary layer accurately attained controlled aspect ratios and y+ values that fully captured near-wall flow with high resolution, a situation quite important for drag predictions. Specific body-of-influence mesh sizing was employed on critical components of interest: the carbox, underbody box, and wakebox, where a high resolution would be required to capture flow separation and reattachment. During the entire simulation, residuals were carefully checked to be less than 10^-4 for continuity, momentum, and turbulence variables to avoid numerical errors. From the stability of important outputs such as drag force, the simulation has reached a state of steady-state convergence. This strict validation of meshing provides the guarantee that the mesh will serve well for analysis of accurate aerodynamic characteristics and effects caused by the augmentation of dimples on drag reduction.

3.3.4. Boundary Condition

Boundary conditions are one of the critical parts of Computational Fluid Dynamics, which is basically how a fluid acts at the limits of the simulation domain and plays an important role in getting accurate and realistic results. The typical kinds of boundary conditions are the inlet conditions, which include velocity inlet, pressure inlet, and mass flow inlet, that dictate the manner of fluid entrance into the domain, while the outlet conditions, such as pressure outlet and velocity outlet, govern the manner of fluid exit. Wall boundary conditions define the interaction of the fluid with solid borders and include the no-slip wall-the velocity of fluid is zero concerning the wall, slip the fluid slides without friction, and moving wall-the wall is in motion.

The following boundary conditions are available in the software: symmetry, periodic, and interface. Symmetry boundary conditions are imposed in problems containing planes of symmetry, reducing the computational domain. Periodic boundary conditions are imposed in periodic flows to allow replication of the domain in space. The interface boundary conditions connect different meshes or regions of the domain in such a way that the transmission of attributes is done correctly. Far-field boundary conditions can represent the infinite extent of the flow domain for problems in external aerodynamics. Initial conditions, though not technically boundary conditions, are required for transient simulations since they define the state of the fluid at the beginning of the simulation.

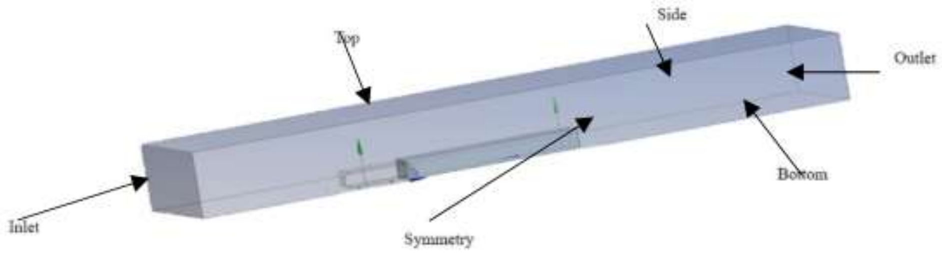


Figure 3.3.4: Boundary conditions of Ahmed's body. (Kumar et al., 2020)

The boundary conditions should be defined through the meshing process in such a way that the solution by the solver is able to accurately detect the boundaries. It involves geometric data creation, mesh formation where subdivision of the domain into pieces takes place, and near the boundary, the mesh resolution must be high. Further steps involve setting the boundary conditions on either the faces or edges of a mesh and defining what kind of boundary condition it is and what kind of value. This involves the importing of a mesh into a CFD solver, verification of boundary conditions, and amendment of specific settings in the solver-like turbulence models or wall functions-to the settings of the geometry. This is accomplished in practice to make sure that a high level of mesh quality is obtained at the boundaries to avoid numerical mistakes, to obtain the resolution required from the boundary layer for flows with viscosity, and also to be consistent with the physical problem being solved to avoid getting non-physical conclusions. The boundary conditions set and imposed during the meshing process lay the very foundation for a successful and accurate CFD simulation.

3.3.5. CFD Calculation

CFD Calculations: This is a process of simulation of the flow of fluids in a structured manner to analyze the model for its aerodynamic characteristics. The process initiates with a model setup through geometry and computational domain definition and by assigning boundary conditions to the model to provide a realistic result. Inlet velocity will be specified next to represent flow entering a system, which acts as a critical prerequisite for the determination of fluid dynamics. The reference area is then computed, usually the projected surface area in the flow direction, to provide a reference for force and coefficient calculation.

Further, reference values include velocity, density, and area, which are set about the conditions of the simulation to ensure that results are correctly normalized. Convergence criteria are defined in the residual setup for numerical accuracy and stability, guiding the solver to the solution with precision. Performance monitoring is done by generating reports for some key aerodynamic parameters: the drag coefficient quantifies resistance relative to the reference area, while the drag force provides the absolute magnitude of resistance experienced by the surface. The process is concluded with initialization, where initial estimates of flow variables are set to enable smooth solver convergence. This structured workflow allows for comprehensive analysis of fluid behavior and aerodynamic efficiency, thus enabling optimization and innovation in engineering applications.

Table 3.3.5: Boundary Condition Ansys Setup

Boundary Condition	Value
Model k-epsilon	2 Eqn
K - epsilon	Standard
Near – Wall Treatment	Standard Wall Functions
CMu	0.09
C1 - Epsilon	1.44
C2 - Epsilon	1.92
TKE Prandtl Number	1
TDK Prandtl Number	1.3
Velocity Magnitude	19.82
Projection Direction	Z
Min Feature Size [m]	0.001
Area [$\boldsymbol{m^2}$]	0.05756999
Density [kg / $\boldsymbol{m^3}$]	1.225
Length [m]	1.044
Temperature [K]	298.15
Velocity [m/s]	19.82
Viscosity [kg/(ms)]	1.837 $\boldsymbol{e^{-5}}$
Ratio of Specific Heats	1.4
Initialization Z Velocity [m / s]	-19.82
Turbulent Kinetic Energy [$\boldsymbol{m^2/s^2}$]	1.473122
Turbulent Dissipation Rate [$\boldsymbol{m^2/s^3}$]	1337.052

a. Model Setup

The first step involves defining the geometry and the computational domain. A model representing the real physical structure under investigation, such as an Ahmed body or other forms of an aerodynamic shape, has to be imported or created. In addition, boundary conditions have to be assigned to the respective regions in the computational domain: inlet, outlet, and wall surfaces. This will ensure that all the parameters are well-set for the simulation environment to accurately represent real-world conditions.

b. Inlet Velocity

$$\mathrm{Re} = \frac{\rho V L}{\mu}$$

$$V = \frac{Re\ \mu}{\rho\ L}$$

$$V = \frac{(1.38\ x\ 10^{6})(1.837\ x\ 10^{-5}\ \frac{Kg}{m}.s)}{(1.225\ kg\ m^{3})(1.044\ m)} = 19.82\ \mathrm{m/s}$$

Temperature: 25°C / 298 K; μ, Dynamic Viscosity: $1.837\ x\ 10^{-5}\ \frac{kg}{m}.s$

Figure 3.3.5.1: Velocity inlet calculation

Inlet velocity is a crucial parameter in studies as it prescribes the nature of flow that enters into the domain. This present step will define the magnitude and direction for the fluid that is going into the system. For example, in flow over the body, this may correspond to some value from experiments or operating conditions necessary to reproduce a particular Reynolds number or any other flow scenario.

c. Compute Area

This normally gives an accurate computation of forces and coefficients, such as drag and lift, by specifying the projected surface area in the direction of fluid flow. It serves as the reference area for the normalization of forces in deriving the dimensionless values, like the drag coefficient.

Reference values serve as a basis for the nondimensionalized analysis by CFD. It would include the reference velocity, temperature, density, area, and other parameters. A correct setting of these ensures that the output results are meaningful in terms of drag coefficient, turbulence intensity, among others, and comparable. The references have to be in harmony with the physical conditions as well as the geometry under usage for the analysis.

d. Residual Setup

The residual setup includes setting up the convergence criteria of the simulation. Residuals are a measure of the numerical error or imbalance in the governing equations solved by the CFD solver. Defining an appropriate threshold for residuals guides the solver to achieve an accurate and stable solution. Monitoring residuals also provides an idea of the progress of the solution during iterations.

e. Drag Coefficient Report Definition

The drag coefficient report is generated to assess the model's aerodynamic performance. This involves defining a monitor or report which would compute the drag coefficient from forces on the surface of the geometry in the direction of the flow of fluid. This dimensionless parameter is relevant for any study related to the effectiveness of design modifications such as the application of dimples.

f. Drag Force Report Definition

Like the drag coefficient, the drag force report refers to the actual force that the fluid exerts on the body. This setup calculates the drag force by integrating the pressure and viscous forces over the surface of the model. It gives the absolute value of resistance that, together with the drag coefficient, can be used to interpret the aerodynamic performance.

g. Initialize

The initialization phase is a precursory step in preparing the computational domain for simulation. Initial guesses of flow variables, such as velocity, pressure, and turbulence quantities, are assigned throughout. These provide starting values for the solver, hence reducing computational time and improving stability. Next, after the initialization, the solution is advanced for a certain number of iterations-say 1000-to converge the solver to an accurate solution. Iterative refinement will ensure reliable outputs such as drag force, drag coefficient, and turbulence kinetic energy, which are of primary importance in the evaluation of the aerodynamic performance and efficiency of a design. By placing such emphasis on proper initialization and iterative calculations, the CFD process achieves robust and precise results.

3.3.6. CFD Post-Processing

Post-processing in computational fluid dynamics involves data analysis from the simulation through specialized software such as Ansys Fluent. Of course, this is a very important stage because it allows meaningful insight to be extracted that guides engineering decisions. This procedure would start with the importation of findings acquired during the preceding computation stage. This makes the operation smooth and burden-free. Further, visualizations and analyses of the results are made to learn different flow parameters like velocity contours, pressure distribution, and characteristics of turbulence.

From here, using Ansys Fluent, one can generate interactive plots with information about the generating flow pattern in the computational domain. The identification of flow patterns is an integral part of the study of fluid mechanics, hence includes scenarios identified as fluid separation, vortices, and regions where fluids circulate in a circular motion. This will also include analysis of quantitative data from simulation results, especially information about forces, moments, and heat transfer rates. That will be further realized with an intricate methodology which enhances the understanding of how the system is working.

Comparative studies compare a simulation's results with real-world data or theoretical predictions that allow for the determination of correctness and reliability regarding a simulation model. The result further extends to a deeper understanding of the root concepts in physics and growing expertise regarding CFD results analysis. Indeed, CFD post-processing allows the creation of needed analytic skills, data visualization, and interpretation-a suite that prepares an individual with relevant knowledge and experience in decisions leading to correct engineering and solving real life-matters pertaining to the realm of fluid dynamics.

3.3.7. Grid Sensitivity Analysis

The Ansys program-Fluid Flow: Fluent-will be employed for grid sensitivity analysis to check the sensitivity of mesh resolution in this simulation. This is a process that deals with the systematic alteration of mesh density while retaining all other parameters constant and appreciating changes in solution accuracy owing to refinement in the mesh. This can first be done by setting up multiple configurations with different levels of refinements, such as from coarse mesh to a finer mesh. Then, simulations are run for each of them, monitoring certain flow parameters such as velocity, pressure, and turbulence in all the resolutions.

The process, iteratively, helps us get useful insight into how changes in mesh resolution result in correctness and convergence of the solution. It also identifies, through differences in solution output among resolutions, the regions where refinement in the mesh is needed for obtaining credible and accurate solutions. Besides, one can analyze computational efficiency by inspecting the required computational resources to run the simulations with different mesh densities. Grid sensitivity analysis allows for the optimization of mesh resolution to achieve a good balance between solution accuracy and computational cost. This, therefore, increases the reliability and performance of CFD simulations.

3.4. Validation Process

The CFD flow methodology for both benchmarking and validation processes is well structured so that it ensures a systematic approach in which simulations will be adequate and uniform. Common stages within the elements are: meshing, quality verification, CAD design, boundary condition configuration, CFD computation, and post-processing. This involves the construction of a computational grid in meshing, while in validation, it looks into specific geometric elements such as Dimple Augmented Ahmed Body Side Design. Validation of quality meshing ensures the high quality of meshes that might prevent numerical errors. The CAD design stage is to construct the accurate geometries for simulation, while the validation process is made up of specialized design elements for scrutiny.

It also involves the proper setting of boundary conditions for both halves with parameters such as input velocities and outlet conditions. The major calculation step involves using software like ANSYS Fluent to solve the numerical values of the governing equations of fluid flow responsible for delivering useful data like flow fields and pressure distributions. Results are analyzed by visualization and explanations in post-processing. The validation covers the boundary layer analysis, turbulence characteristics, and design feature effects in terms of drag reduction, and grid sensitivity analysis that the results are mesh-independent. Both halves include a grid sensitivity analysis, while the approach for validation gives a more critical review to some of the design features.

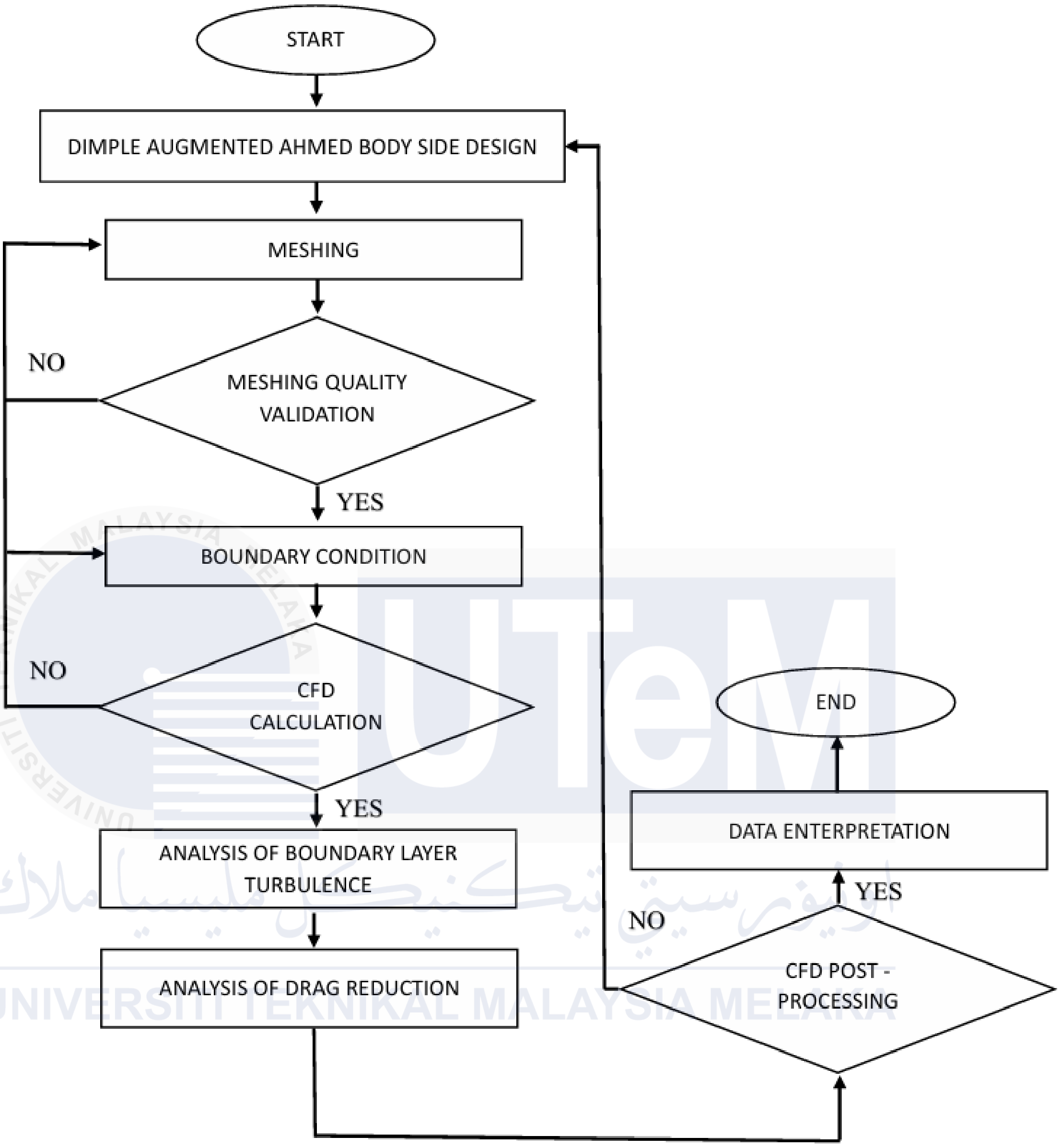


Figure 3.4: Validation flowchart

Figure 8. Validation relies heavily on any study concerning drag reduction. In this part, efficiency concerning design change is assessed by comparing drag forces with or without such features. This step of analysis is highly important for quantifying the advantages of some geometrical changes. Data interpretation is the phase at which inferences are drawn from simulation results and changes to a design are checked, giving more suggestions for optimization. This systematic approach will ensure that CFD simulations are executed in detail

and methodically precise, instructive concerning fluid dynamics and the effectiveness of the design. Dimple Augmented Ahmed Body Side Design

The Dimple Augmented Ahmed Body side design is an improvement of the standard Ahmed Body used in aerodynamics studies to investigate surface modifications affecting fluid dynamics. This geometry design is an improvement developed to carry out studies of drag reduction and turbulence control in case the Ahmed Body carries dimples on its sides. To make a basic shape, this should be done using the CATIA-leading CAD software and with dimensions of 1044mm in length, 288mm in height, and 389mm in breadth. Above are the dimensions that ensure the Ahmed Body dimensions in general use replicate here so that the already benchmarked aerodynamic information could be used to one's favor. The shape of the Ahmed Body includes the placement of spherical dimples at strategic positions engineered to change the characteristics of flow around the vehicle and optimize, perhaps, its aerodynamic performance.

The geometry of the dimple augmentation is designed to keep the same proportions as the Ahmed Body so that the effects measured in aerodynamics are applicable. The dimples are of a size and shape whereby they may effectively modify the flow of air around the model at the same time that it could hold the structural stability of the model. The Ahmed Body surface design has been improved with dimples, in particular, so that the drag would be minimal and the turbulence could be managed at its highest. The final design is then simulated using CFD simulation tests to gain crucial information in relation to improving the aerodynamics even more.

3.4.1. Dimple Augmented Side Design

The dimple designs illustrated in Figure 3.4.1.1 are designed so that boundary layer separation mitigation on a bluff body is enhanced by carefully considering the size and location of the dimples. Precisely, the dimensions used in the study are 313.2 mm, 39.7 mm, 4.95 m², and 4 mm as shown in Table 3.4.1 Concept development in design first involves conceptualization, where theoretical considerations and previous experiment results inform the initial ideas. The ideas are then modeled in CAD to visualize and refine the dimple grid. CFD simulations shall be carried out to predict the performance of the proposed dimple design for a set of flow conditions; this allows iterative optimization to enhance its potential to reduce boundary layer separation.

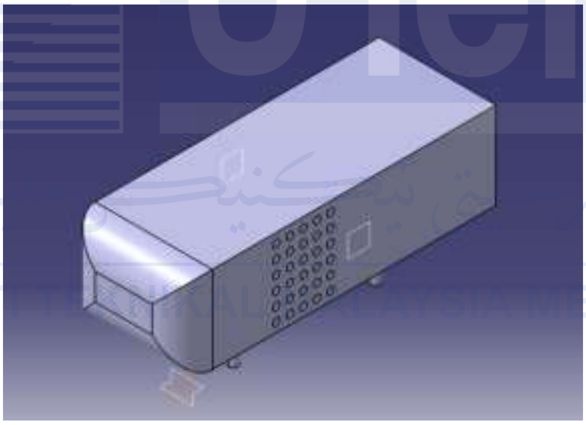


Figure 3.4.1.1: Dimple Augmented Side Design

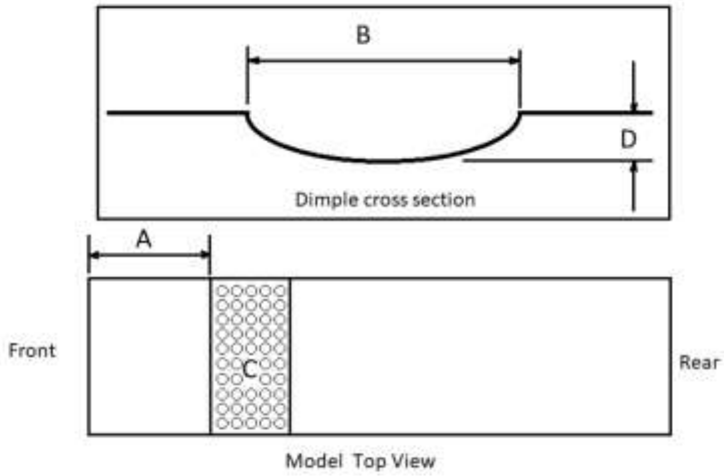


Figure 3.4.1.2: Dimple Specification (Faruq Abdul Latif et al., 2019)

Table 3.4.1: Dimension of Specification

Part	Dimension
A	313.2 mm
B	39.7 mm
C	4.95 m²
D	4 mm

The dimple grid pattern is designed to maximize coverage and optimize spacing so that the dimples, in unison, disturb the boundary layer illustrated in Figure 3.4.1.2. Material selection is crucial concerning durability and manufacturability; thus, the material chosen should withstand forces generated by airflow during testing. Manufacturing includes generating the dimple grid with specified dimensions and geometry, which must be applied on the bluff body surface with good accuracy. The design is tested experimentally by subjecting the bluff body with the dimple grid to airflow under controlled conditions. Performance is measured and compared to a baseline bluff body without dimples to understand the effectiveness of the design. The structured approach for the development and testing of dimples for the reduction of boundary layer separation encourages progress in the field of fluid mechanics and thermal sciences.

3.4.2. Analysis of boundary layer turbulence

The boundary layer turbulence simulation of the dimple augmented Ahmed Body is performed by robust CFD software ANSYS Fluent. The boundary layer is the very area in which, if the properties in fluid flow-in particular, those of velocity and turbulence-change greatly, the determination of performance in aerodynamics will be greatly influenced. In setting up the simulation in ANSYS Fluent, import the geometrical model of the Dimple Augmented Ahmed Body. The already created mesh in CATIA is further enriched so that all subtle characteristics of the dimples are reproduced, and their effects on the boundary layer can also be achieved.

It will thus be necessary to choose turbulence models such as k-ε or k-ω models since this area of flow at the surface of the body represents highly turbulent flow. These are the boundary conditions, correctly set during the simulation so that real flow circumstances may be represented, mainly on input velocity, conditions at the outlet, and those which interact with the walls. ANSYS Fluent further solves for the fluid flow characteristics-velocity, pressure, and turbulence intensity-inside the boundary layer. The obtained data would then be analyzed in order to obtain information about characteristics of turbulence that may be produced by dimples, such as vortex development and smooth-chaotic transition. The analysis is detailed in deducing the effectiveness of the design of dimples to decrease drag and improve aerodynamic efficiency. Optimization analysis would be performed with the aim of improvement in the dimple design, with parameters such as skin friction coefficient and turbulent kinetic energy, to enhance the betterment of the aerodynamics.

3.4.3. Analysis of Drag Reduction

This report presents a study carried out on the aerodynamic performance evaluation of the Dimple Augmented Ahmed Body, supported by ANSYS Fluent software. Analyses are primarily focused on investigating the dimples based on drag reduction ability. The first approach applied for doing the analysis is CFD, where the geometry that included a dimpled one was imported. A refined mesh will enable them to capture flow interactions atop these minute wells-dimples. The intake velocity at the intake, air pressure at the outlet, and surface roughness are some of the simulation parameters that are set based on the faithful reproduction of the actual real-life aerodynamic conditions. Complex turbulent flow over a dimpled surface can be modeled by the application of models such as k-ε or k-ω.

The ANSYS Fluent will eventually provide comprehensive flow data: pressure distribution, the velocity fields, and turbulence intensity. The quantity of drag reduction by the dimples could be quantified by drawing a comparison with the results obtained from Dimple Augmented Ahmed Body with that obtained from a standard Ahmed Body. The key parameters to analyze are the drag coefficient, Cd, which can show how much the aerodynamic drag is reduced. The presence of dimples would break the boundary layer, reduce the pressure drag, and delay the flow separation. All these factors are thus bringing down the overall drag forces. This could enable the engineers to make informed decisions for better improvement in aerodynamic efficiency in the future.

3.4.4. Data interpretation

For the proper study of the effect of dimple augmentation on the aerodynamics of an Ahmed body side, which needs most focuses on Cp and Cd, proper understanding of CFD data is required, where pressure distribution forms the basis for the proper understanding of drag. The pressure contour plots give a good view of the regions of high and low pressure, particularly at the back where the low pressure adds to the drag. Dimples can change the way in which air flows around an object-a fact that sometimes leads to a decrease in resistance due to the formation of favorable pressure gradients. The Cp parametric effectively normalizes the measured pressure in order to help compare pressure gradient over the body. Therefore, it can highlight a place where the dimples would decrease the pressure drag-that is, where the unfavorable pressure gradient would be smoothed and where the flow became uniform.

The drag coefficient (Cd) is a measure of the drag relative to dynamic pressure and reference area, therefore being a direct measure of aerodynamics. A lower coefficient of drag Cd implies better aerodynamic performance, that is, less resistance to the flow. Dimples are able to reduce the coefficient of drag Cd by delaying the flow separation, reducing the size of the wake, and giving rise to an increase in the mixing turbulence. This breaks up the gigantic vortices causing drag, hence drag decreases. This would take a look at the pressure and velocity fields around the Ahmed body, which will indeed reveal the changes to be witnessed in the drag coefficient, Cd. This indeed confirms that all the changes applied to the body will reduce the drag without compromising the stability of the car. The effectiveness of supplementation with dimples for better aero performance can be delved deeper into using CFD, especially as far as Cp and Cd are concerned.

CHAPTER 4: RESULT AND DISCUSSION

4.1 Grid Sensitivity Analysis

Grid sensitivity analysis is one of the most important aspects of CFD simulations. It studies the variation of grid resolution by changing the grid resolution to ensure that the results are independent of grid size. This process gives an idea of where the right balance between computational effort and the improvement in the results is by enhancing the grid and comparing the results. It is ultimately responsible for ensuring the simulation results are reliable and not simply a consequence of the size of the selected grid, which in turn provides confidence in the findings.

4.1.1 Pressure Coefficient (Cp)

Pressure Coefficient (Cp) gives important information about the aerodynamic performance of the benchmark design for various sizes of the grid. It can be identified from the figure that, with an increased size of the grid, the values of Cp are stabilized; this means the results become independent of the grid resolution. This stability reflects the correct capturing of pressure distribution without the influence of grid size in the solution. For example, the 3 million elements Cp values are higher and less refined; it shows that the grid is too coarse to capture detailed pressure variations. The Cp values become consistent at 9 million elements, showing this is a good balance between accuracy and computational cost. Further refinement to 12 and 15 million elements produced minimal changes in Cp values, confirming grid independence.

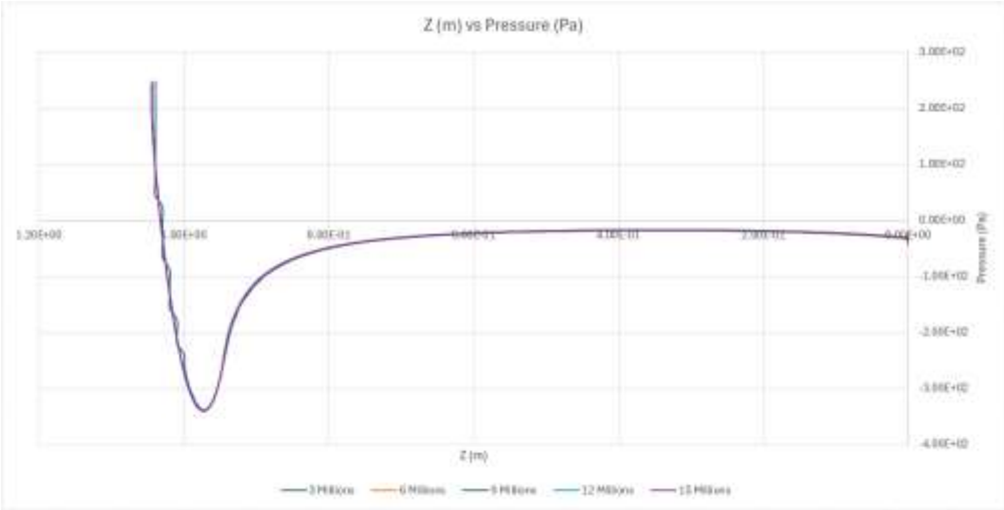


Figure 4.1.1: Pressure Coefficient

Figure 4.1.1 shows that proper grid resolution is very much crucial in capturing the right flow features and hence reliable simulation results. Consistent Cp values at different grid sizes show that the pressure distribution has been correctly captured, which forms the basis of reliable predictions of the aerodynamic performance including drag and lift forces. The graph in general consists of peaks and troughs showing high and low-pressure regions, respectively, which become very important in identifying the areas of high aerodynamic load and possible flow separation. In general, Cp graph analysis shows that the grid size of 9 to 12 million elements is the best compromise for the trade-off between accuracy and computational efficiency.

4.1.2 Comparison of Benchmark Grid Sensitivity Analysis

The Cd Table 4.1.2 compares the drag coefficients for each of the different grid sizes of the CFD simulation. The results in Table 1 show grid sizes ranging from 3 million to 15 million elements, with the corresponding drag coefficient values: 3 million elements-0.3164, 6 million elements-0.3077, 9 million elements-0.3074, 12 million elements-0.3021, and 15 million elements-0.3041. It is possible to notice the drag coefficient decreases with increased grid size, evidencing that finer grids capture more detailed flow features and therefore give more accurate predictions of drag. The most notable reduction in the drag coefficient comes between 3 million and 6 million elements, suggesting that an initial increase in grid resolution has a more significant impact on the accuracy of results. Beyond 9 million elements, the drag coefficient values hardly vary and converge from 0.3074 to 0.3021. The convergence proves that the solution is going to be independent of the grid size, and beyond this value, it will hardly affect the accuracy.

Table 4.1.2: Drag Coefficient Comparison

Number of Elements	Meshing size	Drag Coefficient
3 Million	0.0038	0.3164
6 Million	0.00232	0.3077
9 Million	0.0017	0.3074
12 Million	0.00144	0.3021
15 Million	0.00128	0.3041

Therefore, it will be sufficient to have a grid size of 9 to 12 million elements to get reliable results without wasting any computational resources. The increase in the coefficient of drag for 15 million elements to 0.3041 compared to 0.3021 for 12 million elements could be due to numerical errors or some other settings regarding the simulation. This difference is so minute that it doesn't affect the general trend. This analysis therefore infers that a grid size within the range of 9-12 million elements creates a very good balance between computational efficiency and result accuracy. The drag coefficient table obtained also highlights how sensitive CFD simulations may be toward the size of the grid used, showing that increased grid sizes do improve the accuracy of the drag coefficient results. There are massive improvements up to 9 million elements. Beyond this point, results stabilize, meaning grid independency and, therefore, the best computational efficiency. This analysis ensures that the simulation provides accurate aerodynamic performance with minimum computational cost.

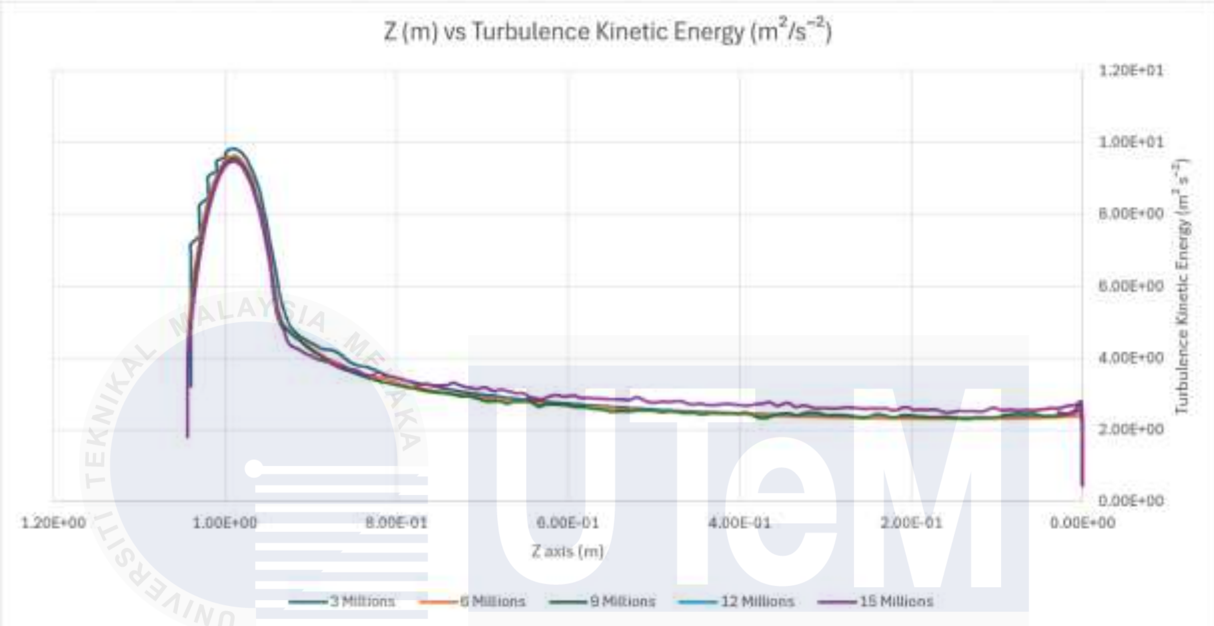


Figure 4.1.2: Turbulence Kinetic Energy

Figure 4.1.2 varies the TKE, from which turbulence characteristics can be inferred for variable grid resolutions. It can be observed that TKE increases with the rise in grid size; or in other words, finer grids capture finer turbulent structures. For example, at 3 million elements, the TKE values are lower, which shows that the coarse grid cannot resolve the finer turbulence structures. Whereas for 6 and 9 million elements, the values of TKE are still growing, showing an increased resolution of turbulence. Finally, for 12 million elements, values of TKE are higher and more consistent, possibly indicating that this grid resolution may be sufficient for resolving the majority of turbulence structures. Beyond this point, at 15 million elements, the values of TKE are rather invariant, reflecting that further grid refinement does not affect the results in a substantial manner.

This turbulence kinetic energy distribution on different planes gives an indication of locations with most prevalent turbulence-a fact very helpful in deducing flow behavior and potentially separated or attached flow. It may well be noticed here that TKE values starting to get stabilized as the size of the grid increases are reflecting thereby results independent of the grid resolution. This indeed indicates a complete stability-suggesting thereby that the simulation truly mimics the nature of turbulence without depending on the grid size. The TKE figure analysis underlines that adequate grid resolution is crucial for the proper capture of turbulence characteristics in order to achieve reliable simulation results. This analysis shows the 12-million-element grid size as the best compromise between the obtained accuracy and computational efficiency.

4.2 Comparison of Dimple and Benchmark Design

This analysis is a comparative study of the aerodynamic efficiency between two different designs: the Dimple Design and the Benchmark Design. The focus of this analysis is on some major parameters like drag coefficient, turbulence kinetic energy (TKE), velocity contours, and pressure distribution. The drag coefficient is the fundamental measure for aerodynamic efficiency, while TKE gives very useful information about the energy contained in turbulent eddies in the flow. Velocity contours allow the visualization of flow characteristics around the model, while the pressure distribution is analyzed using total pressure contours at various planes. The analysis, by investigating these parameters, tries to understand the difference in aerodynamic behavior between the two designs and identify possible ways of improving the performance.

4.2.1 Drag Coefficient

Table 4.2.1 is a comparison between the Cd values for two designs: the Benchmark Design and the Dimple Design. When an object moves through a fluid medium, it experiences a force of resistance, known otherwise as the drag coefficient. The drag coefficient is among the fundamental parameters in the field of aerodynamics. Better aerodynamic efficiency is achieved when the value of Cd is reduced to a lower value.

Table 4.2.1: Drag Coefficient Comparison

Design	Drag Coefficient
Benchmark	0.3021
Dimple Design	0.2790

The present study investigates the drag coefficients for both designs under similar conditions in order to compare the performances of the designs. The results in Table 4.2.1 show that the Benchmark Design has a drag coefficient of 0.3021, whereas the Dimple Design has a lower drag coefficient of 0.2790. Comparison of these values gives an indication that the Dimple Design has reduced drag and increased efficiency, hence serving valuable information concerning the effectiveness of applying dimples to the design. The comparison between the two is relevant for understanding the aerodynamic advantages and potential performance increases of each design.

4.2.2 Velocity Separation

The analysis of the separation zone and the separation velocity demonstrates a considerable difference in aerodynamic efficiency between the reference and dimple configurations. The separation zone characterized with the flow detached from the surface is outlined via pressure coefficient and turbulence kinetic energy contours. As shown in Figure 2.2.1, the benchmark configuration has bigger separation zones with vast regions of low pressure and high turbulence. On the contrary, a dimple configuration shows fewer separation areas, higher pressure values, and lower turbulence intensities, which means the flow has less separation and is more stable, as shown in Figure 2.2.2. This area reduction of separation related to the dimple design represents an increase in aerodynamic efficiency.

Figure 4.2.2.1: Benchmark Separation Region

Figure 4.2.2.2: Dimple Design Separation Region

The Velocity separation phenomenon is investigated using velocity contours with velocity fluctuations within the separation region as shown in Figure 4.2.2.1 and 4.2.2.2. In comparison, the reference design shows large discrepancies in velocity close to the separation area, hence indicating a more unstable flow and higher drag. On the other hand, the dimple design displays smoother velocity contours with less apparent velocity separation, hence it is promoting more streamlined and stable flow. This uniformity in dimple design creates lower drag and improved aerodynamic performance according to Figure 4.2.2.5.

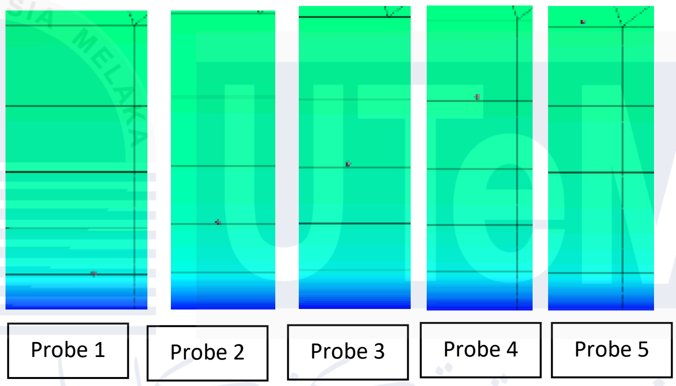


Figure 4.2.2.3: Benchmark Separation Probe Position

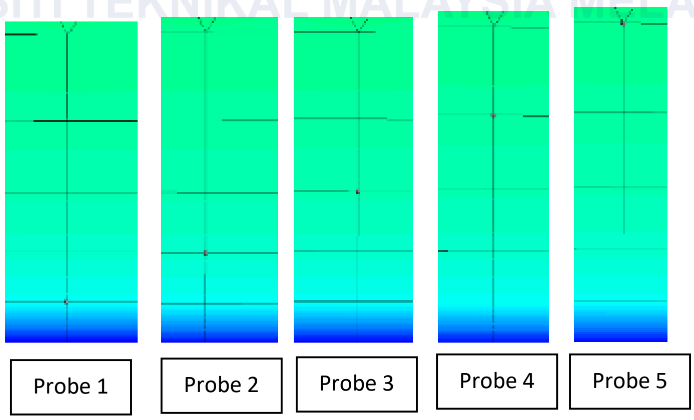


Figure 4.2.2.4: Dimple Design Separation Probe Position

Table 4.2.2: Velocity Separation Difference

Velocity Separation	Benchmark	Dimple Design
Probe 1	9.41663	8.87538
Probe 2	11.3384	10.7891
Probe 3	12.5101	11.9479
Probe 4	13.2712	12.7891
Probe 5	13.8923	13.7241

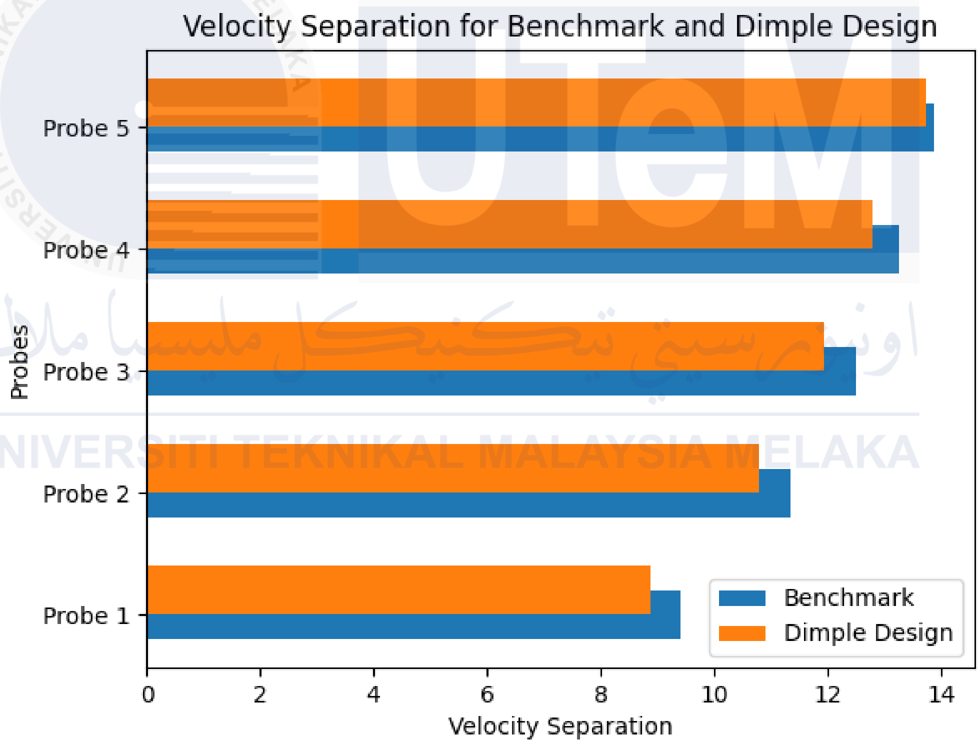


Figure 4.2.2.5: Velocity Separation Bar Graph

Overall, the dimple design's ability to minimize both the separation region and velocity separation highlights its superior aerodynamic efficiency and stability compared to the benchmark design.

4.2.3 Effect of Velocity Distribution

Knowledge of the velocity distribution in a flow field enables the determination of the aerodynamic performance and stability of a design. Regions of the flow where the velocity gradients are large are more susceptible to flow separation, where the flow lifts off the surface, leaving a wake and increasing drag. On the other hand, smoother velocity distributions help to maintain the flow attached, therefore reducing drag and increasing efficiency. An evenly distributed velocity field also ensures uniform distribution of pressure, which is substantiated in Appendix C, one that is necessary for both maintaining aerodynamic stability and good overall performance.

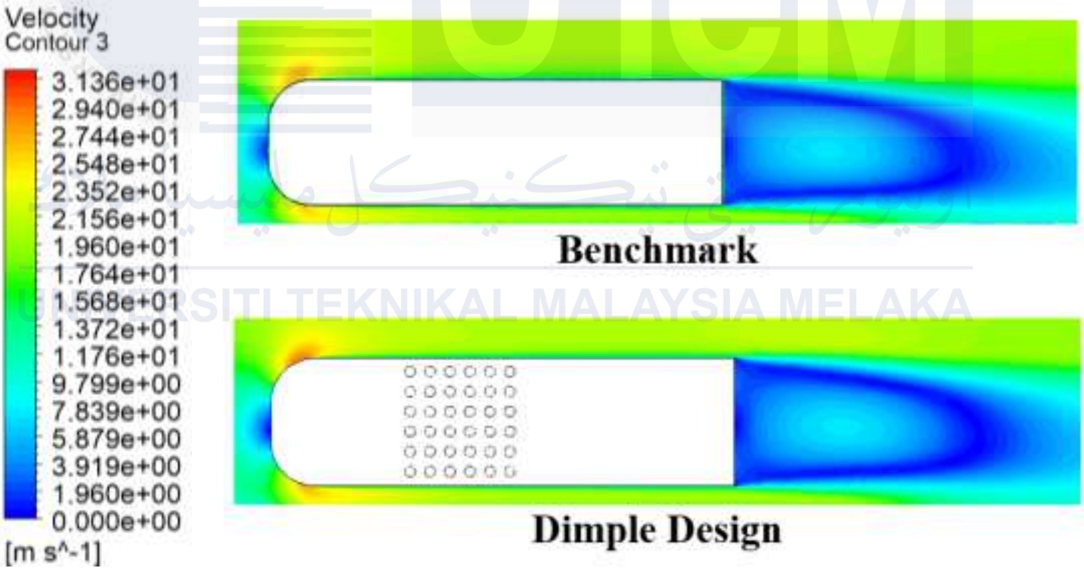


Figure 4.2.3: Velocity Contour

Furthermore, the velocity profile shown in Figure 4.2.3 has a strong influence on the level of turbulence in the flow. High velocity gradients can lead to high levels of turbulence, which increase energy dissipation and reduce flow efficiency. Configurations with more gradual velocity profiles usually have lower levels of turbulence; hence, there is less energy dissipation and better aerodynamic efficiency. This happens because differences in velocity distribution have a large impact on drag and lift aerodynamic forces on any design taken into consideration. A non-uniform pressure field can develop since the velocity distribution is not uniform, hence leading to an increase in drag and reduction in lift. As such, effective distribution of velocity becomes one of the most relevant factors to improve aerodynamic performance and decrease drag while maintaining or increasing lift. The understanding of the velocity distribution and its optimization is, therefore, the major concerns in developing efficient and stable aerodynamic systems.

4.2.4 Pressure Distribution

The pressure distribution in the entire flow field is the prime factor in determining the aerodynamic efficiency of any given design because of its direct relation with lift and drag forces experienced on the body. Figure 4.2.4 shows that the pressure coefficient, Cp, is contour-plotting the pressure variation along the surface. Regions of high and low pressure are represented by elevated and reduced Cp values, respectively. In the original design case, there is great variation shown in the contours of Cp, with quite extensive low-pressure regions to potentially provide flow separation zones. In the dimple configuration, Cp contours are quite stable and uniform, as shown by the more homogeneous distribution of pressure and a significant rise in Cp values in regions where the benchmark configuration shows low pressure. This indicates delayed flow separation and a more stable flow.

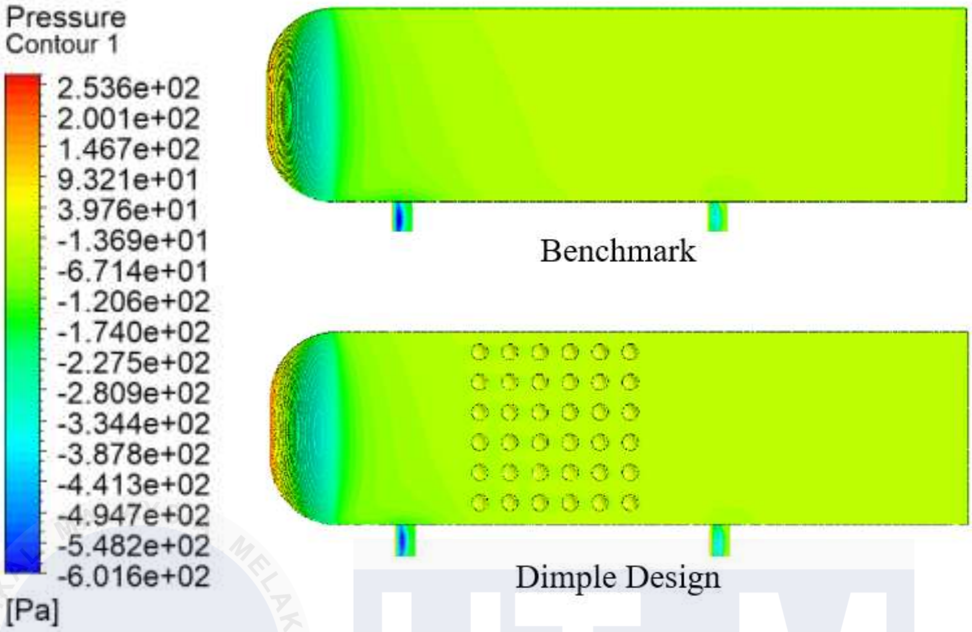


Figure 4.2.4: Pressure Contour

The benchmark design depicts a non-uniform pressure distribution and, hence, high drag and even potential flow instability. On the other hand, the dimple design retains a more uniform pressure distribution; lower drag is obtained, and generally, better overall aerodynamic performance. The high values of Cp for the dimple design indicate a better pressure recovery and, hence, less flow separation—in general, this projects increased stability and efficiency, according to Appendix D and Appendix E. In conclusion, the analysis of pressure distribution confirms the great importance of surface design to aerodynamic efficiency. More precisely, the dimples configuration shows better behaviour because of an enhanced pressure distribution.

In reality, it is important to achieve an ideally distributed pressure field for maintaining aerodynamic stability and hence high performance. High-pressure zones act in favour of providing lift, whereas low-pressure areas can only increase drag if not attended to accordingly. Whereas it may be intuitively conceived that various surface features, for example, provide a closer-to-even distribution of pressure that works to reduce drag and therefore greatly improve the efficiency of the whole. And so, this balance of high and low pressures is rooted in the principle of drag reduction and stability in objects' airflow. Thus, understanding how pressure diffuses and knowing how to manipulate this has been at the very heart of the development of flight systems that fly and stay stable.

4.3 Turbulence Kinetic Energy

The TKE analysis epitomes how turbulence dynamics coupled with the Bernoulli Principle can be used to optimize aerodynamics. As suggested by the Bernoulli Principle, with an increase in the velocity of a fluid, pressure becomes reduced. This is accomplished because the dimple augmentation on an Ahmed body minimizes energy loss in flow through reductions in TKE and results in an overall stabilized velocity distribution along with a streamlined flow pattern.

This stabilization has a direct consequence on the pressure distribution: with smoother and quicker flow, the zones of high pressure associated with flow separation and recirculation are reduced. Aligning the velocity and pressure gradients better, drag forces decrease, hence increasing aerodynamic efficiency. Furthermore, the dimples regulate turbulent energy for easier flow reattachment and reduce pressure fluctuations along the body surface. This synergy between TKE reduction and Bernoulli's velocity-pressure relationship proves how effective the dimple design has been in producing significant drag reduction along with improved flow dynamics.

4.3.1 TKE Distribution across Planes

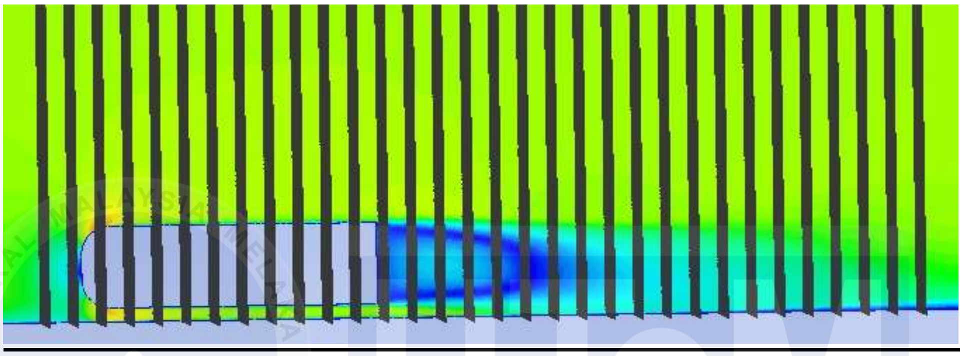


Figure 4.3.1.1: Plane position, Dimple Design at Plane 4 to Plane 7

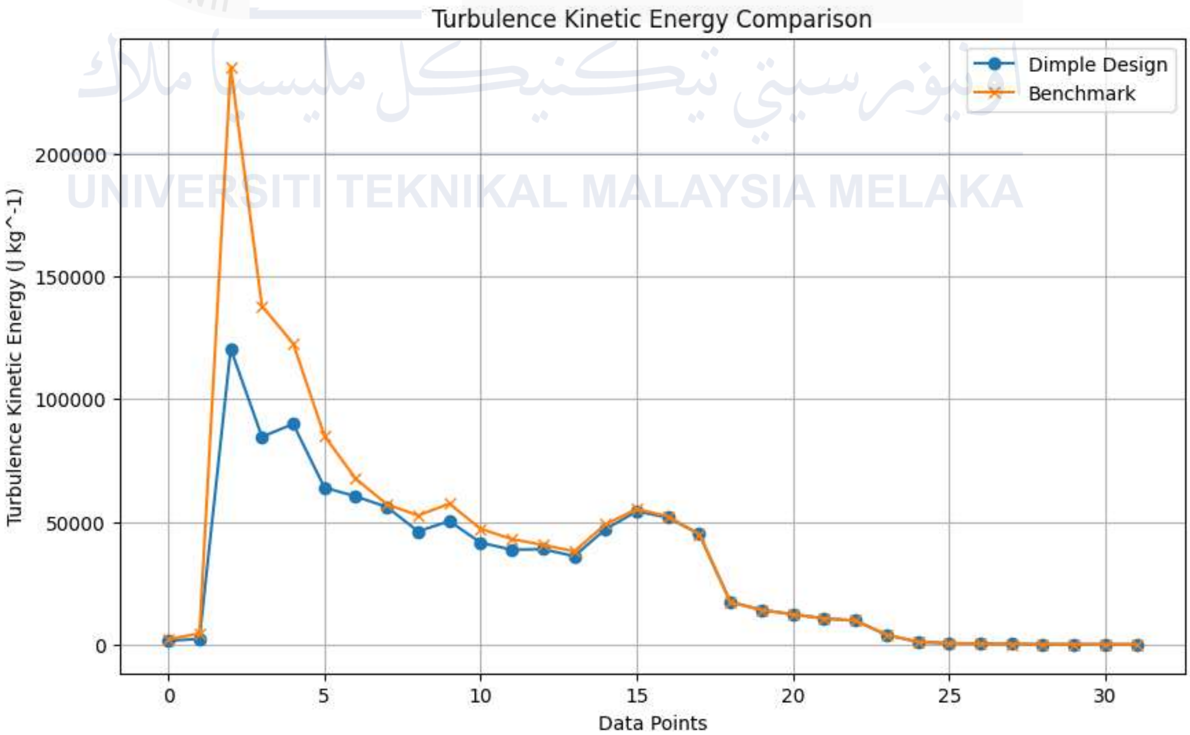


Figure 4.3.1.2: TKE Graph by plane

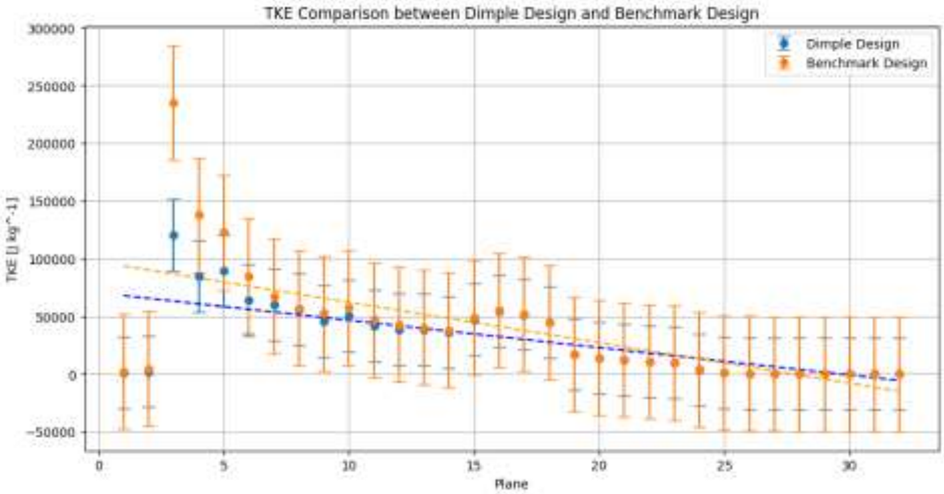


Figure 4.3.1.3: Error bar and Trend line of Graph

Plane-by-plane analysis of TKE sheds much light on the aerodynamic performances between the benchmark and the dimple design. Values of TKE were computed over 32 planes all 0.1 meters apart from front to rear of an Ahmed body as shown below in Figure 4.3.1.1. Figures 4.3.1.2 and 4.3.1.3 presents the maximum TKE values for both designs at plane 3: the benchmark design reaches a value of 1.205e5 J/kg and the dimple design at 2.352e5 J/kg. That means this position is a significant turbulence region. In general, from the TKE plot, it is possible to observe that the dimple design has a higher value compared to the benchmark design, which could point to more turbulence induced by the dimple design. Besides, the TKE values for both designs tend to go down when moving toward the rear part of the Ahmed body, which indicates a reduction in turbulence. The above analysis will provide an all-rounded understanding of the characteristics of turbulence in both designs while pointing out the influence of the dimple design in increasing the turbulence levels.

4.3.2 Mean Turbulence Kinetic Energy Comparison

Table 4.3.2: Mean Comparison

Design	Mean (J/kg)
Dimple	3.12 e 4
Benchmark	3.95 e 4

The mean value of TKE represents the average turbulence kinetic energy across all planes. It can be seen that the mean TKE value for the dimple design is higher than that for the benchmark design, which shows that on average, the dimple design induces more turbulence. This could be due to the surface modifications of the dimple design that enhance turbulent flow.

4.3.3 Median TKE Values Analysis

Table 4.3.3: Median Comparison

Design	Median (J/kg)
Dimple	2.66 e 4
Benchmark	2.76 e 4

The median value is defined as the TKE value that stands in the middle if all values of TKE were to be arranged in increasing order. It is also a measure of central tendency not as badly affected by extreme values. Indeed, the higher median TKE value obtained by the dimple design verifies observations that generally it produced higher levels of turbulence compared to the benchmark design.

4.3.4 Standard Deviation of TKE Values

Table 4.3.4: Standard Deviation Comparison

Design	Median (J/Kg)
Dimple	3.10 e 4
Benchmark	4.97 e 4

The standard deviation describes the dispersion or variability of the magnitude of TKE. The higher standard deviation in the dimple design shows that there is more variation in the level of turbulence across different planes. This would, therefore, mean that the dimple design has been able to generate more fluctuating patterns of turbulence compared to the benchmark design.

4.3.5 Statistical Significance: T-Test Results

Table 4.3.5: T-Test Result

Analysis	Value
T Test	-0.782
P Value	0.427

Results from the T-Test prove that the dimple enhancement on the Ahmed body does not contribute to any statistically significant drag reduction compared to the benchmark design. A t-statistic of -0.782 obtained and the P-value of 0.427 show that the difference in averages between the two groups is not statistically significant and can be due to chance.

As such, the T-Test implies that the dimple design is not significantly improved in terms of its aerodynamics compared to the benchmark design. Because the drag-reducing performance increase with dimple augmentation does not provide statistically significant improvement, and differences found are also not enough to make valid claims of outperforming a benchmark design regarding drag reduction.

4.3.6 Wilcoxon Test Results

Table 4.3.6: Wilcoxon Test Result

Analysis	Value
W statistic	98.5
P Value	0.0013

The following output of the Wilcoxon test of the experiment shows that there is a statistically significant difference in the Dimple Design and Benchmark energy values. The W is 98.5, while the P-value for that is 0.0013, indicating that these differences could barely be due to random chance.

The P-value is very low, less than the common significance threshold of 0.05; thus, the drag on an Ahmed body with dimple enhancement is significantly lower compared to the reference design. For this reason, according to the Wilcoxon test, the dimple design has an aerodynamic performance that is statistically significant over the benchmark design. This suggests that the application of dimples to the Ahmed body significantly reduces drag and increases its overall aerodynamic efficiency.

4.4 TKE Contour Graph

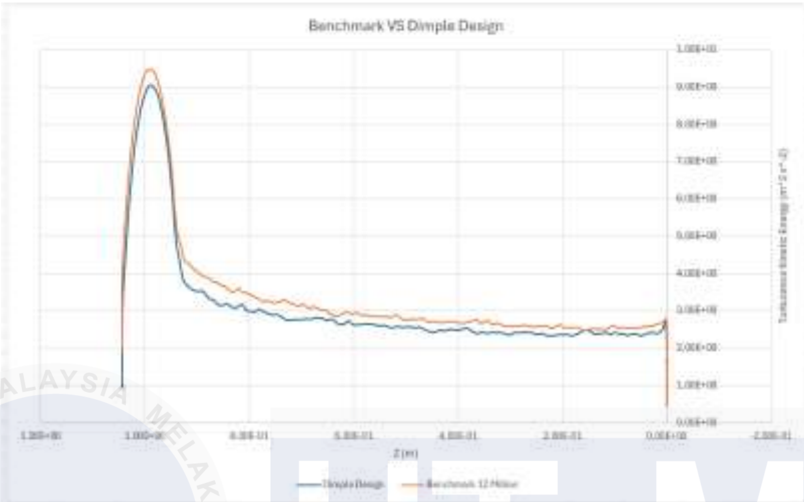


Figure 4.4: TKE Contour

Figure 4.4 gives in detail the turbulence distribution that is necessary for understanding and further optimization of its aerodynamic performance. Regions of high TKE, usually formed around obstacles or any gradient area, would denote areas of intense turbulence. These are areas that may contribute to increased drag and hence energy losses. Low TKE denotes low turbulence, indicative of smooth flow, and is often found in streamlined areas or areas far away from obstacles, indicating good aerodynamics.

The dimple design has always shown lower TKE values compared to the benchmark design for all planes. From this reduction in turbulence, it is expected that the drag is reduced, hence, overall performance improvement. A lower TKE value in the dimple design shows better management of airflow around it, hence less dissipation of energy and more stability.

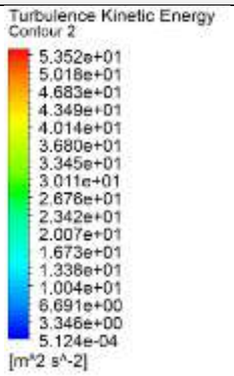
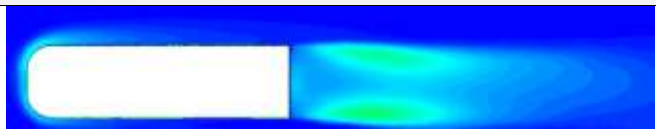
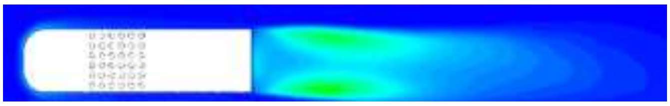
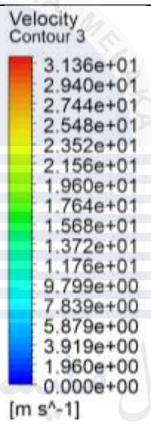
The mesh size also influences the contour graph of TKE. The employment of finer meshes will give a much finer, more accurate description of TKE distribution, thereby enabling the capturing of the smaller turbulent structures that may not be seen in coarser meshes. This resolution becomes critical for detailed analysis and optimization, as one gets a better insight into the flow behaviour to the instance where turbulence can, up to a certain extent, be minimized.

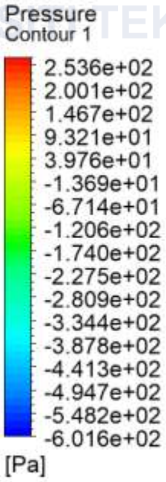
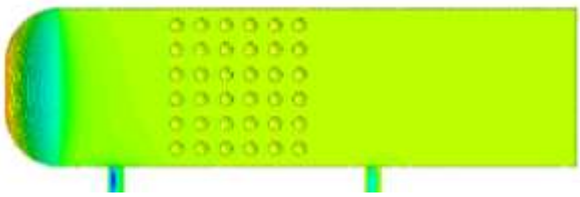
In general, from the TKE contour graph, the dimple design has evidenced its superiority in terms of better aerodynamic advantages compared to the benchmark design. This shows that a detailed mesh analysis is very essential in order to understand the nuances in turbulence distribution. Comparing visualization of TKE values, it allows the engineers to draw appropriate inferences regarding design feature optimization, drag reduction, and performance improvement for moving ahead with better aerodynamic efficiency towards superior design outcomes.

4.5 Plane Contour Table

Table 3.3 presents the comparison of three important parameters, pressure, velocity, and TKE for different planes, which are important in order to understand the designs from an aerodynamic performance and flow characteristics point of view.

Table 4.5: Contour Table

<table>
<tr><th>Contour</th><th>Value</th><th>Figure</th></tr>
<tr><td>Turbulence
Kinetic
Energy</td><td>
</td><td>
Benchmark

Dimple Design</td></tr>
<tr><td>Velocity</td><td>
</td><td>
Benchmark

Dimple Design</td></tr>
<tr><td>Pressure</td><td>
</td><td>
Benchmark

Dimple Design</td></tr>
</table>

4.5.1 Pressure

Pressure contours clearly outline areas of high pressure, which are normally around leading edges or points of stagnation, indicating strong aerodynamic forces on the surface. These areas are crucial in terms of load distribution. Low-pressure areas are typically where the flow is accelerated and are responsible for much of the lift that is crucial to overall performance and stability of a design.

4.5.2 Velocity

The velocity distribution reveals areas of flow acceleration and deceleration. High-velocity regions, typically due to narrowing passages or streamlined surfaces, correspond to lower pressure according to Bernoulli's principle, which is beneficial for lift generation. Low-velocity regions, found in areas of flow separation or recirculation, can lead to higher pressure and potential flow instability, affecting aerodynamic performance.

4.5.3 Turbulence Kinetic Energy (TKE)

The TKE values indicate the level of turbulence in the flow. High TKE areas are those of high turbulence, normally around obstacles or high-velocity gradients, which result in increased drag and losses of energy. Low TKE areas suggest that the flow is smoother, with less turbulence, and therefore more efficient aerodynamically. Comparing the dimple design to the benchmark design, it can be seen that the dimple design has consistently lower TKE values, highlighting its superior aerodynamic efficiency.

4.6 Limitation

The study finds several limitations that could affect generalization and the applicability of the findings. The Ahmed body is a reduced model of real geometries of vehicles and, therefore, may not capture all the detailed flow dynamics for real vehicle geometries, which perhaps limits the finding's applicability to real situations. These CFD simulations have been carried out with various assumptions and approximations, like turbulence models and boundary conditions, which might be wrong or biased. Turbulence models have a very strong influence on the precision of flow properties and drag reduction predictions.

All experimental verification has been done under controlled conditions that are not representative of on-road driving, variable wind conditions, irregularities in road surfaces, and vehicle interactions that may affect the actual aerodynamic performance and drag reduction, which have not been adequately considered to date.

The investigation focused on specific dimple configurations and their positions on the Ahmed body. While these showed promising results, the scope of this work was narrow; hence, the examination of other dimple patterns and different positions is not considered. Further research is required to establish the optimal geometries for a wide range of vehicle shapes and sizes.

Experiments and simulations were conducted at a certain scale that may or may not correctly represent full-scale vehicles. Scale effects can arise both in flow behavior and in aerodynamic performance and usually require painstaking care and validation when results are scaled up. In this study, only one flow condition was focused on, which generally represents a steady-state situation. In reality, however, driving involves fluctuating speeds and yaw angles with transient situations. Besides, this study did not deal with material aspects and manufacturing issues associated with applying the dimple design to real automobiles. These are of much practical concern for real-life applications in material durability, production feasibility, and cost-effectiveness but beyond the scope of the present study.

CHAPTER 5: CONCLUSION

5.1 Conclusion

This work has widely studied the application of dimples on the lateral surfaces of the Ahmed body to reduce TKE and drag. The effort of this study was to change the flow dynamics over the Ahmed body by introducing dimples with an intention of enhancing its aerodynamic performance. In such a context, the subsequent analysis showed that the introduction of dimples indeed altered the TKE distribution in a way that caused local reductions in turbulence. This modification allowed a more stable attachment of the boundary layer, which reduced the creation of wake vortices and therefore reduced the aerodynamic drag.

All the above results were verified using a rigorous grid sensitivity analysis for reliability. A validation study confirmed that the CFD simulations were robust, high-resolution meshing, and state-of-the-art turbulence modeling techniques produced consistent and accurate results. The efficacy of this validation thus strengthened the credibility of the observed drag reduction effects due to augmentation by dimples.

A strong correlation was established between the drag reduction and the modulation of TKE. The dimples favored a cohesively flowing flow which reduced turbulence and substantially improved the aerodynamic efficiency of the Ahmed body. This relationship indicates that management of TKE plays a great role in successful drag reduction and also underlines surface modification as one of the feasible methods for passive control in automobile design.

In brief, the increased dimples on Ahmed body surfaces have already demonstrated very impressive potential for drag reduction and flow characteristic optimization. Based on this, it may also be implied that strategic changes to vehicle surfaces bear very promising results for vehicle performance, fuel economy, and, above all, environmental sustainability. The different types of dimples are configured further in these proposals as an extension to develop a finer resolution of optimization in drag.

5.2 Recommendation

These findings from the study provide a number of recommendations for better understanding and application of dimple augmentation for drag reduction in vehicle aerodynamics. The future research work should be extended from a simple Ahmed body model to more complex and realistic vehicle geometries that can help explain, under practical conditions, the effects of dimple augmentation and thus provide more applicable results for automotive industries. The improvement in the accuracy of CFD simulations can be achieved by refining turbulence models and boundary conditions. Advanced models of turbulence, validated against experimental data, reduce errors and biases, thus providing better predictions of the aerodynamic performance. Empirical trials for enhancing CFD simulations have to be performed under various driving situations, including a wide range of speeds, yaw angles, and transient states; thus, providing a more complete understanding of the efficacy of dimple augmentation in practical applications.

Further studies should be performed with a wider range of geometrical parameters, including size, shape, and position of the dimples, to find out the optimal dimple geometry for various vehicle types and flow conditions to gain maximum benefits due to drag reduction. It needs to consider scale effects when extrapolating experimental and simulation findings in the present research to full-size vehicles by testing at different scales and formulating scaling equations that can predict the effectiveness of dimple augmentation in practical applications with better accuracy. From the practical point of view of dimple designs, investigation of material properties, durability, and manufacturing limits is needed; thus, research must be focused on appropriate material and cheap manufacturing methods in order to ensure feasibility for dimple augmentation in mass production.

Therefore, the use of dimple augmentation has presented a number of complex issues to be resolved collaboratively by the aerodynamicists, material scientists, and automobile engineers. It calls for interdepartmental research, innovative ideas, and speedy implementation in automobile design. Further research developed from these findings should enhance these findings from this study in further improving and developing greener and more sustainable vehicle designs through the use of dimple augmentation.

REFERENCES

Ahmed, W. S. R., Ramm, ; G, & Faltin, G. (n.d.). *E: Tectlnical Paper Series 840300 Some Salient Features Of The Ti me-Ave rage d G rou n d Vehi cle*.

Akhter, M. Z., Ali, A. R., & Omar, F. K. (2022). Aerodynamics of a three-dimensional bionic morphing flap. *SUSTAINABLE ENERGY TECHNOLOGIES AND ASSESSMENTS*, *52*(D). https://doi.org/10.1016/j.seta.2022.102286

Arksey, H., & O'Malley, L. (2005). Scoping studies: Towards a methodological framework. *International Journal of Social Research Methodology: Theory and Practice*, *8*(1), 19–32. https://doi.org/10.1080/1364557032000119616

Bonnavion, G., Borée, J., & Herbert, V. (2022). On the use of kinetic energy balance for the volumetric identification of drag sources of a blunt body. Application to road vehicles. *International Journal of Heat and Fluid Flow*, *96*. https://doi.org/10.1016/j.ijheatfluidflow.2022.108977

Booysen, A., Das, P., & Ghaemi, S. (2022). Large-scale 3D-PTV measurement of Ahmed-body wake in crossflow. *Experimental Thermal and Fluid Science*, *132*. https://doi.org/10.1016/j.expthermflusci.2021.110562

Cafiero, G., Amico, E., & Iuso, G. (2024). Manipulation of a turbulent boundary layer using sinusoidal riblets. *JOURNAL OF FLUID MECHANICS*, *984*. https://doi.org/10.1017/jfm.2024.256

Chen, G., Li, X., Zhang, L., Liang, X., Meng, S., & Zhou, D. (2022). Numerical analysis of the effect of train length on train aerodynamicperformance. *AIP ADVANCES*, *12*(2). https://doi.org/10.1063/5.0079587

Chen, X., Zhong, S., Ozer, O., Kennaugh, A., Liu, T., & Gao, G. (2024). A Study of spatiotemporal features of sweeping jets acting on afterbody vortices using low-operation-rate stereo PIV. *Experimental Thermal and Fluid Science*, *158*. https://doi.org/10.1016/j.expthermflusci.2024.111244

Essel, E., Das, S., & Balachandar, R. (2020). Effects of rear angle on the turbulentwake flow between two in-line ahmed bodies. *Atmosphere*, *11*(4). https://doi.org/10.3390/atmos11040328

Faruq Abdul Latif, M., Nur Othman, M., Fairuz Zahmani, Q., Safwa Khashi, N., Eik Zhen, B., Farid Ismail, M., & Yusuf Ismail, A. (2019). Optimization of Boundary Layer Separation Reduction Induced by The Addition of a Dimple Grid on Top of a Bluff Body. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences Journal Homepage: Open Access Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, *64*(2), 173–182. www.akademiabaru.com/arfmts.html

Fowler, T. S., Witherden, F. D., & Girimaji, S. S. (2020). Partially-averaged Navier-Stokes simulations of turbulent flow past a square cylinder: Comparative assessment of statistics and coherent structures at different resolutions. *Physics of Fluids*, *32*(12). https://doi.org/10.1063/5.0027590

Gattere, F., Zanolini, M., Gatti, D., Bernardini, M., & Quadrio, M. (2024). Turbulent drag reduction with streamwise-travelling waves in thecompressible regime. *JOURNAL OF FLUID MECHANICS*, *987*. https://doi.org/10.1017/jfm.2024.408

Greenhalgh, T., Thorne, S., & Malterud, K. (2018). Time to challenge the spurious hierarchy of systematic over narrative reviews? *European Journal of Clinical Investigation*, *48*(6). https://doi.org/10.1111/eci.12931

Hamada, S., Yakeno, A., & Obayashi, S. (2023). Drag reduction effect of distributed roughness on the transitional flowstate using direct numerical simulation. *INTERNATIONAL JOURNAL OF HEAT AND FLUID FLOW*, *104*. https://doi.org/10.1016/j.ijheatfluidflow.2023.109230

Kamble, C., & Girimaji, S. (2020). Characterization of coherent structures in turbulent wake of a sphere using partially averaged Navier-Stokes (PANS) simulations. *Physics of Fluids*, *32*(10). https://doi.org/10.1063/5.0024854

Kamble, C., Girimaji, S. S., & Chen, H.-C. (2020). Partially averaged navier–stokes formulation of a two-layer turbulence model. *AIAA Journal*, *58*(1), 174 – 183. https://doi.org/10.2514/1.J058742

Koppa Shivanna, N., Ranjan, P., & Clement, S. (2021). The effect of rear cavity modifications on the drag and flow fieldtopology of a Square Back Ahmed Body. *PROCEEDINGS OF THE INSTITUTION OF MECHANICAL ENGINEERS PART D-JOURNAL OFAUTOMOBILE ENGINEERING*, *235*(7), 1849–1863.

https://doi.org/10.1177/0954407021990918

Kumar, G., Mboreha, C. A., & Al-fakih, Y. (2020). Numerical simulations on the aerodynamics of the Ahmed body at different slant angles. *International Journal of Advanced Engineering Research and Applications*, *6*(03), 55–71. https://doi.org/10.46593/ijaera.2020.v06i03.002

Lai, C., Fu, H., Hu, B., Ling, Z., & Jiang, L. (2020). Aerodynamic drag reduction and optimization of MIRA model based on plasma actuator. *Actuators*, *9*(3). https://doi.org/10.3390/ACT9030064

Lehmkuhl, O., Houzeaux, G., Owen, H., Chrysokentis, G., & Rodriguez, I. (2019). A low-dissipation finite element scheme for scale resolving simulations of turbulent flows. *Journal of Computational Physics*, *390*, 51 – 65. https://doi.org/10.1016/j.jcp.2019.04.004

Li, S., Luo, Z., Deng, X., Peng, W., & Liu, Z. (2021). Experimental investigation on active control of flow around afinite-length square cylinder using dual synthetic jet. *JOURNAL OF WIND ENGINEERING AND INDUSTRIAL AERODYNAMICS*, *210*. https://doi.org/10.1016/j.jweia.2021.104519

Liberati, A., Altman, D. G., Tetzlaff, J., Mulrow, C., Gøtzsche, P. C., Ioannidis, J. P., Clarke, M., Devereaux, P., Kleijnen, J., & Moher, D. (n.d.). *The PRISMA Statement for Reporting Systematic Reviews and Meta-Analyses of Studies That Evaluate Health Care Interventions: Explanation and Elaboration*. www.annals.org

Lin, C.-Y., Owolabi, B. E., & Lin, C.-A. (2022). Polymer-turbulence interactions in a complex flow and implications forthe drag reduction phenomenon. *PHYSICS OF FLUIDS*, *34*(4). https://doi.org/10.1063/5.0086686

Liu, J., Zhang, Z., Wang, Y., Li, M., & Xu, S. (2021). Experimental study on the three-dimensional wake structure of the DrivAer standard model. *Journal of Visualization*, *24*(3), 443 – 460. https://doi.org/10.1007/s12650-020-00714-2

Lloyd, C. J., Mittal, K., Dutta, S., Dorrell, R. M., Peakall, J., Keevil, G. M., & Burns, A. D. (2023). Multi-fidelity modelling of shark skin denticle flows: insights intodrag generation mechanisms. *ROYAL SOCIETY OPEN SCIENCE*, *10*(2). https://doi.org/10.1098/rsos.220684

Luciano, T. (2011). Doing a literature review: releasing the social science research imagination. *Evaluation & Research in Education*, *24*(4), 303–304. https://doi.org/10.1080/09500790.2011.588012

Luo, J., Mi, K., Tan, D., Zhang, Z., Li, M., Qing, J., & Huang, H. (2022). Investigation of the Aerodynamic Characteristics of Platoon Vehicles Based on Ahmed Body. *Shock and Vibration*, *2022*. https://doi.org/10.1155/2022/3269604

Maulenkul, S., Yerzhanov, K., Kabidollayev, A., Kamalov, B., Batay, S., Zhao, Y., & Wei, D. (2021). An arbitrary hybrid turbulence modeling approach for efficient and accurate automotive aerodynamic analysis and design optimization. *Fluids*, *6*(11). https://doi.org/10.3390/fluids6110407

McDermott, M., Resende, P., Charpentier, T., Wilson, M., Afonso, A., Harbottle, D., & de Boer, G. (2020). A FENE-P *k* - ε Viscoelastic Turbulence Model Valid up to HighDrag Reduction without Friction Velocity Dependence. *APPLIED SCIENCES-BASEL*, *10*(22). https://doi.org/10.3390/app10228140

McDermott, M., Resende, P. R., Wilson, M. C. T., Afonso A. M.and Harbottle, D., & de Boer, G. (2021). An improved k-ω turbulence model for FENE-P fluids without frictionvelocity dependence. *INTERNATIONAL JOURNAL OF HEAT AND FLUID FLOW*, *90*. https://doi.org/10.1016/j.ijheatfluidflow.2021.108799

Mohammadikalakoo, B., Schito, P., & Mani, M. (2020). Passive flow control on Ahmed body by rear linking tunnels. *Journal of Wind Engineering and Industrial Aerodynamics*, *205*. https://doi.org/10.1016/j.jweia.2020.104330

Moher, D., Liberati, A., Tetzlaff, J., Altman, D. G., Antes, G., Atkins, D., Barbour, V., Barrowman, N., Berlin, J. A., Clark, J., Clarke, M., Cook, D., D'Amico, R., Deeks, J. J., Devereaux, P. J., Dickersin, K., Egger, M., Ernst, E., Gøtzsche, P. C., … Tugwell, P. (2009). Preferred reporting items for systematic reviews and meta-analyses: The PRISMA statement. In *PLoS Medicine* (Vol. 6, Issue 7). Public Library of Science. https://doi.org/10.1371/journal.pmed.1000097

Mondal, A., Chatterjee, S., Tariang, A. M., Raj, L. P., & Debnath, K. (2023). Numerical investigation of on-demand fluidic winglet aerodynamicperformance and turbulent characterization of a low aspect ratio wing. *ADVANCES IN AIRCRAFT AND*

SPACECRAFT SCIENCE, *10*(2), 107–125. https://doi.org/10.12989/aas.2023.10.2.107

Page, M. J., McKenzie, J. E., Bossuyt, P. M., Boutron, I., Hoffmann, T. C., Mulrow, C. D., Shamseer, L., Tetzlaff, J. M., Akl, E. A., Brennan, S. E., Chou, R., Glanville, J., Grimshaw, J. M., Hróbjartsson, A., Lalu, M. M., Li, T., Loder, E. W., Mayo-Wilson, E., McDonald, S., … Moher, D. (2021). The PRISMA 2020 statement: An updated guideline for reporting systematic reviews. *Journal of Clinical Epidemiology*, *134*, 178–189. https://doi.org/10.1016/j.jclinepi.2021.03.001

Pearson, F. (2014). Systematic approaches to a successful literature review. *Educational Psychology in Practice*, *30*(2), 205–206. https://doi.org/10.1080/02667363.2014.900913

Podvin, B., Pellerin, S., Fraigneau, Y., Evrard, A., & Cadot, O. (2020). Proper orthogonal decomposition analysis and modelling of the wake deviation behind a squareback Ahmed body. *Physical Review Fluids*, *5*(6). https://doi.org/10.1103/PhysRevFluids.5.064612

Rao, C., & Liu, H. (2020). Effects of Reynolds Number and Distribution on Passive Flow Control inOwl-Inspired Leading-Edge Serrations. *INTEGRATIVE AND COMPARATIVE BIOLOGY*, *60*(5), 1135–1146. https://doi.org/10.1093/icb/icaa119

Ricco, P., Skote, M., & Leschziner, M. A. (2021). A review of turbulent skin-friction drag reduction by near-walltransverse forcing. *PROGRESS IN AEROSPACE SCIENCES*, *123*. https://doi.org/10.1016/j.paerosci.2021.100713

Serafini, F., Battista, F., Gualtieri, P., & Casciola, C. M. (2024). Kinetic Energy Budget in Turbulent Flows of Dilute Polymer Solutions. *FLOW TURBULENCE AND COMBUSTION*, *112*(1, SI), 3–14. https://doi.org/10.1007/s10494-023-00460-z

Shea, B. J., Reeves, B. C., Wells, G., Thuku, M., Hamel, C., Moran, J., Moher, D., Tugwell, P., Welch, V., Kristjansson, E., & Henry, D. A. (2017). AMSTAR 2: A critical appraisal tool for systematic reviews that include randomised or non-randomised studies of healthcare interventions, or both. *BMJ (Online)*, *358*. https://doi.org/10.1136/bmj.j4008

Siddiqui, N. A., & Chaab, M. A. (2021). A Simple Passive Device for the Drag Reduction of an Ahmed Body. *Journal of Applied Fluid Mechanics*, *14*(1), 147 – 164. https://doi.org/10.47176/jafm.14.01.31791

Systematic Reviews In The Social Sciences_ A Practical Guide By Mark Petticrew. (n.d.).

Tranfield, D., Denyer, D., & Smart, P. (n.d.). *Towards a Methodology for Developing Evidence-Informed Management Knowledge by Means of Systematic Review **.

Tricco, A. C., Lillie, E., Zarin, W., O'Brien, K. K., Colquhoun, H., Levac, D., Moher, D., Peters, M. D. J., Horsley, T., Weeks, L., Hempel, S., Akl, E. A., Chang, C., McGowan, J., Stewart, L., Hartling, L., Aldcroft, A., Wilson, M. G., Garritty, C., … Straus, S. E. (2018). PRISMA extension for scoping reviews (PRISMA-ScR): Checklist and explanation. In *Annals of Internal Medicine* (Vol. 169, Issue 7, pp. 467–473). American College of Physicians. https://doi.org/10.7326/M18-0850

Verma, M. K., Alam, S., & Chatterjee, S. (2020). Turbulent drag reduction in magnetohydrodynamic and quasi-staticmagnetohydrodynamic turbulence. *PHYSICS OF PLASMAS*, *27*(5). https://doi.org/10.1063/1.5142294

Viswanathan, H. (2021). Aerodynamic performance of several passive vortex generator configurations on an Ahmed body subjected to yaw angles. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, *43*(3), 1–23. https://doi.org/10.1007/s40430-021-02850-8

Wang, J., Minelli, G., Dong, T., He, K., & Krajnovic, S. (2020). Impact of the bogies and cavities on the aerodynamic behaviour of ahigh-speed train. An IDDES study. *JOURNAL OF WIND ENGINEERING AND INDUSTRIAL AERODYNAMICS*, *207*. https://doi.org/10.1016/j.jweia.2020.104406

Wang, L., Lu, H., Tian, Z., Yang, Y., Guo Shuangand Wang, H., & Kong, X. (2022). Numerical Study of the Ratio of Depth-to-Print Diameter on thePerformance and Flow Characteristics for a Dimpled, Highly LoadedCompressor Cascade. *AEROSPACE*, *9*(8). https://doi.org/10.3390/aerospace9080422

Webster, J., & Watson, R. T. (2002). Analyzing the Past to Prepare for the Future: Writing a Literature Review. In *Source: MIS Quarterly* (Vol. 26, Issue 2).

Yagmur, S., Dogan, S., Aksoy, M. H., & Goktepeli, I. (2020). Turbulence modeling approaches on unsteady flow structures around a semi-circular cylinder. *Ocean Engineering*, *200*. https://doi.org/10.1016/j.oceaneng.2020.107051

Yang, B., Peng, C., Wang, G., & Wang, L.-P. (2021). A direct numerical simulation study of flow modulation and turbulentsedimentation in particle-laden downward channel flows. *PHYSICS OF FLUIDS*, *33*(9). https://doi.org/10.1063/5.0062017

Yousefi, A., Niazi Ardekani, M., & Brandt, L. (2020). Modulation of turbulence by finite-size particles in statisticallysteady-state homogeneous shear turbulence. *JOURNAL OF FLUID MECHANICS*, *899*. https://doi.org/10.1017/jfm.2020.457

Yu, Z., Ping, H., Liu, X., Zhu, H., Wang, R., Bao, Y., Zhou, D., Han, Z., & Xu, H. (2020). Turbulent wake suppression of circular cylinder flow by two smallcounter-rotating rods. *PHYSICS OF FLUIDS*, *32*(11). https://doi.org/10.1063/5.0023881

Yudianto, A., Adiyasa, I. W., & Yudantoko, A. (2021). Aerodynamics of bus platooning under crosswind. *Automotive Experiences*, *4*(3), 119 – 130. https://doi.org/10.31603/ae.5298

Yudianto, A., Sofyan, H., Gunadi, & Fauzi, N. A. (2022). Aerodynamic Characteristics of Overtaking Bus under Crosswind: CFD Investigation. *CFD Letters*, *14*(8), 20 – 32. https://doi.org/10.37934/cfdl.14.8.2032

Zhai, H., Jiao, D., & Zhou, H. (2023). Parametric Optimization of Wheel Spoke Structure for Drag Reduction of an Ahmed Body. *CMES - Computer Modeling in Engineering and Sciences*, *139*(1), 955 – 975. https://doi.org/10.32604/cmes.2023.043322

Zhang, Y., Ye, Z., Li, B., Xie, L., Zou, J., & Zheng, Y. (2022). Numerical analysis of turbulence characteristics in a flat-plate flowwith riblets control. *ADVANCES IN AERODYNAMICS*, *4*(1). https://doi.org/10.1186/s42774-022-00115-z

APPENDIX

Appendix A: Gaant Chart of PSM 1

No.	Task	Location	Week													
			1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Module on PSM 1	Online														
2	Explaination of the topic and confirmation by SV	SV's Room														
3	References finding using SLR Methods	Technician Room														
4	Download all references	Technician Room														
5	Module on Literature Review	Lecture Hall 1														
6	Drafting Chapter 2	Technician Room														
7	Module on Methodology	Lecture Hall 1														
8	Chapter 2 Process: Sub Topic Explaination	SV's Room														
9	Chapter 2 Submission & Chapter 3 Flowchart drafting	SV's Room														
10	Flowchart Checking	Online														
11	Chapter 1 & 3 writing															
12	Chapter 1 & 3 writing															
13	Chapter 3 checking	SV's Room														
14	PSM 1 report submission															

Appendix B: Gaant Chart of PSM 2

No.	Task	Location	Week													
			1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	PSM 2 Briefing	Lecture Hall 1														
2	Running Dimple Design analysis	Studio Analysis Automotive														
3	Data gathering for Dimple Design Analysis	Studio Analysis Automotive														
4	Data gathering for Dimple Design Analysis	Studio Analysis Automotive														
5	BDP Workshop Series: AI Tools in Thesis Writing	Lecture Hall 1														
6	BDP Workshop Series: Statistical Data Analysis for BDP	Lecture Hall 1														
7	Talent Enhancement 2024	Lecture Hall 1														
8	BDP Workshop Series: Do's and Don’ts in writing result and discussion	Lecture Hall 1														
9	Chapter 4 checking	SV’s Room														
10	Report Writing															
11	PSM 2 Report Submission to Supervisor															

12	Report Submission to Panel															
13	Presentation															

Appendix C: Velocity Distribution

Plane	Dimple Design [m s^-1]	Benchmark [m s^-1]
1	1.677e+5	1.725e+5
2	1.776e+5	1.812e+5
3	6.783e+5	6.702e+5
4	5.366e+5	5.343e+5
5	4.922e+5	4.886e+5
6	4.712e+5	4.713e+5
7	4.551e+5	4.619e+5
8	4.466e+5	4.540e+5
9	4.458e+5	4.541e+5
10	4.319e+5	4.420e+5
11	4.507e+5	4.585e+5
12	4.573e+5	4.607e+5
13	4.727e+5	4.587e+5
14	2.067e+5	2.116e+5
15	2.156e+5	2.181e+5
16	2.204e+5	2.266e+5
17	2.242e+5	2.266e+5
18	2.256e+5	2.278e+5
19	1.641e+5	1.677e+5
20	1.652e+5	1.669e+5
21	1.692e+5	1.715e+5
22	1.685e+5	1.716e+5
23	1.664e+5	1.732e+5
24	9.206e+4	9.679e+4
25	4.472e+4	4.809e+4
26	3.087e+4	3.335e+4
27	2.346e+4	2.744e+4
28	1.904e+4	2.381e+4
29	1.568e+4	2.137e+4
30	1.388e+4	1.896e+4
31	1.190e+4	1.784e+4
32	1.074e+4	1.692e+4

Appendix D: Pressure Distribution on Vertical Plane

Plane	Dimple Design (Pa)	Benchmark (Pa)
2	4.356e+5	4.236e+5
3	-3.479e+6	-3.433e+6
4	-1.980e+6	-2.109e+6
5	-1.410e+6	-1.531e+6
6	-8.383e+5	-9.114e+5
7	-5.784e+5	-6.898e+5
8	-4.389e+5	-5.597e+5
9	-4.140e+5	-4.783e+5
10	-5.048e+5	-5.758e+5
11	-5.067e+5	-5.799e+5
12	-5.904e+5	-6.522e+5
13	-8.438e+5	-8.552e+5

Appendix E: Pressure Distribution on Horizontal Plane

Plane	Dimple Design (Pa)	Benchmark (Pa)
Plane 33	-4.621e+6	-4.621e+6
Plane 34	-1.214e+6	-1.214e+6
Plane 35	-1.051e+6	-1.051e+6
Plane 36	-1.003e+6	-1.003e+6
Plane 37	-1.062e+6	-1.062e+6
Plane 38	-1.715e+6	-1.715e+6
Plane 39	-5.270e+6	-5.270e+6

Appendix F: TKE Contour Plane

Plane	Dimple Design [J kg^-1]	Benchmark [J kg^-1]
1	1.531e+3	2.200e+3
2	2.345e+3	4.498e+3
3	1.205e+5	2.352e+5
4	8.461e+4	1.379e+5
5	8.983e+4	1.226e+5
6	6.396e+4	8.501e+4
7	6.035e+4	6.758e+4
8	5.602e+4	5.713e+4
9	4.611e+4	5.257e+4
10	5.028e+4	5.736e+4
11	4.149e+4	4.716e+4
12	3.863e+4	4.295e+4
13	3.886e+4	4.057e+4
14	3.598e+4	3.797e+4
15	4.703e+4	4.900e+4
16	5.425e+4	5.522e+4
17	5.172e+4	5.219e+4
18	4.500e+4	4.494e+4
19	1.733e+4	1.716e+4
20	1.409e+4	1.412e+4
21	1.235e+4	1.223e+4
22	1.064e+4	1.065e+4
23	9.776e+3	9.746e+3
24	3.906e+3	4.053e+3
25	1.129e+3	1.079e+3
26	5.846e+2	5.224e+2
27	3.749e+2	3.229e+2
28	2.388e+2	2.017e+2
29	1.617e+2	1.461e+2
30	1.360e+2	1.102e+2
31	1.202e+2	9.517e+1
32	9.585e+1	8.425e+1