FACULTY OF MECHANICAL TECHNOLOGY AND ENGINEERING

INVESTIGATING THE INFLUENCE OF DIMPLES ON SIDE OF AHMED BODY SKIN FRICTION USING COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

SHAFIQ DANIAL BIN KAMARULLZAMAN

B092110171

BACHELOR OF MECHANICAL ENGINEERING TECHNOLOGY (AUTOMOTIVE) WITH HONOURS

2025

Faculty of Mechanical Technology and Engineering

INVESTIGATING THE INFLUENCE OF DIMPLES ON SIDE OF AHMED BODY SKIN FRICTION USING COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

Shafiq Danial Bin Kamarullzaman

Bachelor of Mechanical Engineering Technology (Automotive) with Honours

2025

INVESTIGATING THE INFLUENCE OF DIMPLES ON SIDE OF AHMED BODY SKIN FRICTION USING COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

SHAFIQ DANIAL BIN KAMARULLZAMAN

A thesis submitted in fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering Technology (Automotive) with Honours

Faculty Of Mechanical Technology and Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2025

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **INVESTIGATING THE INFLUENCE OF DIMPLES ON SIDE OF AHMED BODY SKIN FRICTION USING COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS**

SESI PENGAJIAN: **2023-2024 Semester 1**

Saya   : **SHAFIQ DANIAL BIN KAMARULLZAMAN**

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: _____6th January 2025_____   Tarikh: _____10th 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. entitled "Investigating the Influence of Dimples on Side
of Ahmed Body Side Skin Friction Using Computational Fluid Dynamics (CFD) Analysis"
is the result of my own research except as cited in the references. The Choose an item. has
not been accepted for any degree and is not concurrently submitted in candidature of any
other degree.

Signature   :

Name   :   SHAFIQ DANIAL BIN KAMARULLZAMAN

Date   :   6th 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 Engineering Technology (Automotive) with Honours.

Signature   :

Supervisor Name   : TS. MOHD FARUQ BIN ABDUL LATIF

Date   : 10th January 2025

DEDICATION

This thesis is dedicated to:

The sake of Allah, My Creator and My Master.

My beloved family, thank you so much for not losing hope, for your endless support through thick and thin, and for your unstoppable prayer and blessing throughout my studies.

My supervisor, Ts. Mohd Faruq Bin Abdul Latif, thank you for all the knowledge, guidance, support, and advice that inspires me to gain more knowledge. May Allah bless you always.

Lastly, thank you to all my friends who always support each other. Wishing you all good luck in your career and life

ABSTRACT

Aerodynamic drag significantly contributes to vehicle performance, fuel efficiency, and ecological friendliness; hence, it is the most important field in auto-body engineering. Despite the overwhelming research in Ahmed bodies, the amenability of side dimples to drag and skin friction remains unexploited. In this regard, this work investigates the effect of side dimples on the aerodynamics of the Ahmed body using CFD methods. The steps that have been carried out systematically were 3D modeling, mesh generation, setting of boundary conditions, and CFD analyses. The number of elements identified for mesh sensitivity analysis was 12 million, as this number had the best fitting where the computational cost and accuracy were well balanced. It was observed that the drag coefficient decreased by 8.46% from its benchmark value of 0.306 to 0.277 in this modified model, while the skin friction coefficient slightly reduced from 0.014 to 0.0135. Statistical analysis and pressure distribution comparisons supported the stability of the modified model aerodynamics, with negligible deviation over the benchmark configuration. Lift and moment coefficients after their initial transients stabilized into a consistent value of aerodynamics. This finding confirms that side dimples act to increase the aerodynamic performance by reducing flow detachment and hence turbulence. Such modifications provide practical ways in which the fuel economy could be improved and emissions reduced, thus having economic and environmental benefits. The research forms a basis for new and innovative optimization in aerodynamics toward achieving sustainable engineering solutions in the automotive sector.

ABSTRAK

Rintangan aerodinamik secara signifikan menyumbang kepada prestasi kenderaan, kecekapan bahan api, dan kecekapan ekologi; oleh itu, ia merupakan bidang paling penting dalam kejuruteraan badan automotif. Walaupun terdapat banyak kajian mengenai badan Ahmed, kesan lekukan sisi terhadap rintangan dan geseran permukaan masih belum diterokai. Dalam hal ini, kajian ini menyiasat kesan lekukan sisi terhadap aerodinamik Ahmed body menggunakan kaedah CFD. Langkah-langkah yang dijalankan secara sistematik termasuk pemodelan 3D, penjanaan mesh, penetapan syarat sempadan, dan analisis CFD. Jumlah elemen yang dikenal pasti untuk analisis sensitiviti mesh adalah 12 juta, kerana bilangan ini memberikan keseimbangan terbaik antara kos pengiraan dan ketepatan. Pemerhatian menunjukkan bahawa pekali rintangan menurun sebanyak 8.46%, daripada nilai penanda aras 0.306 kepada 0.277 dalam model yang diubah suai, manakala pekali geseran permukaan sedikit menurun daripada 0.014 kepada 0.0135. Analisis statistik dan perbandingan taburan tekanan menyokong kestabilan aerodinamik model yang diubah suai, dengan penyimpangan yang sangat kecil berbanding konfigurasi penanda aras. Pekali angkat dan momen menunjukkan kestabilan selepas gangguan awal, mencerminkan prestasi aerodinamik yang konsisten. Penemuan ini mengesahkan bahawa lekukan sisi meningkatkan prestasi aerodinamik dengan mengurangkan pemisahan aliran dan seterusnya mengurangkan pergolakan. Pengubahsuaian seperti ini menyediakan cara praktikal untuk meningkatkan kecekapan bahan api dan mengurangkan pelepasan, memberikan manfaat ekonomi dan alam sekitar. Kajian ini menjadi asas kepada pengoptimuman aerodinamik yang inovatif ke arah mencapai penyelesaian kejuruteraan lestari dalam sektor automotif.

ACKNOWLEDGEMENTS

First of all, I would like to show my appreciation to my supervisor, Ts. Mohd Faruq Bin Abdul Latif, for the endless support he gave throughout completing my Bachelor's Degree report. Despite his hectic schedule, he managed to spare some time to meet with me and provide helpful information and advice. All credit for this research goes to him because, without his kind assistance, this thesis would not have been completed on time.

Besides that, I would like to express my gratitude to my parents, Encik Kamarullzaman Bin Abd Latiff and Puan Zarinah Binti Abu Bakar for giving their full support mentally, spiritually, and financially. Without them, I may never have continued my studies and overcome the obstacles on the long journey of my studies. In addition, they provided me with financial support throughout the research.

Finally, I would like to thank everyone who directly or indirectly helped and guided me to complete this thesis. Their advice and encouragement enabled me to overcome the obstacles in this project. May Allah bless them for their contribution and effort.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS AND ABBREVIATIONS

k	-	Turbulent kinetic energy
t	-	Time (s)
x_i, x_j	-	Spatial coordinates (m)
u_i, u_j	-	Velocity components (m/s)
ρ	-	Density (kg/m³)
p	-	Pressure (Pa)
μ	-	Dynamic viscosity (Pa·s or N·s/m²)
ϵ	-	Turbulent dissipation rate (m²/s³)
μ_t	-	Turbulent viscosity (Pa·s)
f_i	-	Time-averaged pressure
P_k	-	Production of turbulent kinetic energy (m²/s³)
C_1, C_2	-	Empirical constants (dimensionless)
M	-	Pitching moment (N·m)
S	-	Reference area (m²)
c	-	Mean aerodynamic chord (m)
L	-	Lift force (N)
D	-	Drag force (N)
τ_w	-	Wall shear stress (N/m²)
C_m	-	Coefficient of moment
C_d	-	Coefficient of drag
C_l	-	Coefficient of lift
C_f	-	Skin friction coefficient

LIST OF APPENDICES

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

CHAPTER 1

INTRODUCTION

1.1   Research Background

Aerodynamic drag is one of the most crucial factors within the automobile industry. It directly links to the performance of a vehicle along with fuel efficiency and environmental impact. The skin friction drag is an integral aerodynamic drag component that results directly from the surface characteristics and fluid viscosity. The Ahmed body is an idealized model developed by S.R. Ahmed that finds wide use in experimental aerodynamics because it permits duplication of the relevant parts of an automobile's airflow. So, focused studies can be carried out over the aerodynamic phenomena (Ahmed, 1984).

This would effectively reduce drag by changing the characteristic of the boundary layer and delaying the separation of flow with the help of surface modifications, such as dimples, because the two influence the golf balls (Kim & Choi, 2006). While plenty of experiments have been carried out to date on bettering aerodynamics with the Ahmed body, there is an appreciable shortage regarding the room where side dimples have their effect. To cater to this shortage, the current study is aimed at employing advanced Computational Fluid Dynamics techniques to study the impacts of side dimples on reducing the friction and drag of the Ahmed body.

The hypothesis is that the presence of lateral dimples will reduce aerodynamic drag and increase overall flow effectiveness. Therefore, the design will make use of new inventive ideas to maximize fuel economy and performance, leading to major economic and

environmental benefits. The systematic approach incorporates thorough CFD simulations, model setup, mesh generation, boundary condition description, simulation run, and output analysis. The results obtained are cross-checked with literature to ensure correctness and reliability. The output from the study will be of far-reaching importance to academic and applied research on aerodynamic optimization.

1.2   Problem Statement

There are very many potential cases for reduction in aerodynamic drag in the automotive industry, specifically for improved fuel efficiency and performance. Therefore, Ahmed body, as a simplified vehicle model, became one of the most important benchmarks in the presented problems related to the possibility of representation of the main features of the flow. No research is available regarding drag reduction by the side surface modification, which incorporates the side surfaces with dimples in the present case on the Ahmed body. Therefore, a lack of proper investigations limits data on the effect of side dimples on skin friction and drag, with wide chances of missing valuable opportunities for better aerodynamics. Instead, the gap should be bridged due to advancements of drag reduction strategies offering great economic and environmental benefits, in this case, reducing fuel consumption and emissions. The present study attempts to investigate the effect of the side dimples on an Ahmed body through the analysis of Computational Fluid Dynamics (CFD), offering some insights that could be useful in guiding some creative design approaches in the automotive area.

1.3 Research Question

By translating the problem statement into a research question, the main research has been stated as follows:

RQ1 : How will the side dimples on Ahmed Body affect the skin friction with respect to a smooth surface?

RQ2 : How will the side dimples affect the overall value for drag coefficient for the Ahmed body?

RQ3 : How do the side dimples impact the skin friction drag of the Ahmed Body?

1.4   Research Objective

This work hopes to contribute valuable data about the effect of side dimples on the aerodynamic performance of the Ahmed body, in particular, skin friction reduction and overall drag. Numerical studies using the CFD technique will be done, and the following specific objectives are pursued:

   i.   Quantify the effect of side dimples on skin friction drag for an Ahmed body using Computational Fluid Dynamics.

   ii.   Evaluate the effect of side dimples on the total drag coefficient of an Ahmed body.

   iii.   Investigating the impact of side dimples on the Ahmed body's pressure distribution using CFD analysis for improved aerodynamic performance.

1.5   Research Scope

The research work will deeply investigate the role played by side dimples in improving the aerodynamic performance of Ahmed body in vehicle aerodynamics. It

basically investigates the surface modification of the Ahmed body, majorly the case of side dimples, as the basic geometric model. The project systematically examines variations in dimple attributes and arrangement, to assess their influence on the most fundamental aerodynamic elements.

The primary parameters that would be investigated are the skin friction drag, and the overall drag coefficient that the software would provide in great detail through fully developed CFD simulations. The computational methodology will be validated thoroughly by comparing the results obtained with the available experimental data or previous computational results, if available.

The study mainly focuses on the Ahmed body for insights that could have an influence on design methodologies for various vehicle applications in the automotive industry. Limitation of time, computational resources, and expertise will define the scope of research. This is to ensure that the methodology followed will be achievable and that the resources are used effectively.

1.6   Expected Result

This research contributes important insight into the aerodynamic performance of an Ahmed body with side dimples. Friction drag should be further diminished in the presence of side dimples, and the reduction varies to a great extent with the characteristics that can boost the technology-based developments of side dimples on a large scale. These dimple enhancements would affect the aerodynamic efficiency and would bring the drag of the Ahmed body down by amount.

The aim of the present research study is to establish the improved dimple configurations that provide improved drag reduction through customized design features

using parametric research upon drag reduction and offering knowledge upon the aerodynamic principles involving the drag reduction from dimples for enhanced comprehension on better flow control strategies.

Validation of the CFD simulation based on experimental data or prior studies will prove the credibility of the methodology. The results, therefore, promise to affect the application of real-world automotive aerodynamics in such a way that the introduction of innovative methods is allowed, leading to the improvement of the performance and fuel and environment friendliness. The envisaged results can point out the potential for better comprehension and application of modifications in the given field of aerodynamics with regard to vehicles.

1.7   Thesis Structure

This thesis investigates the impact of side dimples on Ahmed Body Aerodynamics through five meticulously structured chapters aimed at achieving certain objectives. Chapter 1, the Introduction, starts by stating the research's purpose, providing background information on aerodynamic drag reduction, and presenting the study's logic, objectives, and assumptions.

Following that, the Literature Review provides an in-depth overview of aerodynamic drag and its elements, highlighting the significance of the Ahmed body as a benchmark model. The article reviews previous research on surface modifications, namely dimples, highlighting the limitations of current studies and identifying areas for further investigation and research challenges.

Chapter 3 will describe how this research is carried out. It discusses the geometry of the Ahmed body, computational setup, and the Computational Fluid Dynamics (CFD)

analysis method, encompassing turbulence models and solver setups. The document outlines criteria for dimple variations and validation processes for CFD simulations and offers insights into data processing methodologies.

Furthermore, Research Foundation and Analysis in Chapter 4 presents findings from CFD simulations, such as skin friction drag, total drag coefficient, and the influence of dimple designs on aerodynamic performance. The most effective dimple designs for lowering drag are discovered through comprehensive analysis and comparison with validation data.

The final chapters evaluate the results to the research objectives, contrasting them with existing literature and theoretical predictions. The article discusses the implications of the results for vehicle aerodynamic design, lists the study's limitations, and offers recommendations for future research endeavours. This chapter examines the limitations seen when using side dimples to Ahmed Body Aerodynamics and suggests possible remedies. The study's conclusions are useful for improving and advancing aerodynamic design approaches in the future.

CHAPTER 2

LITERATURE REVIEW

2.1   Preliminaries

This chapter thoroughly examines the essential elements required for the study. The study begins by systematically detailing and explaining the importance of terms such as Dimples, Ahmed Body, Side Skin Friction, Drag Reduction, and Computational Fluid Dynamics (CFD) Analysis. The chapter aims to provide a strong foundational understanding of key terms to aid in interpreting subsequent subjects by offering clear and precise definitions.

This research explores the historical context of the investigation, providing explanations for the development of present-day scientific discussions and the earliest research in the study area. The chapter evaluates the method implemented in previous research, comparing the modeling tools used and their benefits and drawbacks. This chapter critically examines the method utilized in this research, thus laying the foundation for how the technique will be employed and evaluated and thus allowing an all-inclusive understanding of the research strategy.

2.2   Systematic Literature Review (SLR)

Systematic reviews are critical for synthesizing information and require a methodology that is as rigorous as that of primary research (Higgins & Green, 2011). They

have a role to play with respect to creating, disseminating, and ensuring appropriate information to support evidence-based decisions and practices and are adamant in being transparent and replicable (Moher et al., 2009). Systematic reviews are different from nonsystematic reviews in that they are explicit and transparent in their methods and follow predetermined steps in an ordered way aimed at reducing bias and increasing reliability (Liberati et al., 2009). Stringent adherence to this diligent recombinant ensures that their output represents a gold standard of methodology in the academic conversation, whereby knowledge can be advanced across a number of fields (Higgins et al., 2019).

Systematic reviews substantially guide the decision-making process beyond academic boundaries. Combining multiple research sources in integrated stories gives policymakers useful insights based on the most reliable evidence (Gough et al., 2017). The mutual relationship between these variables highlights their crucial role in developing evidence-based policies and in providing more effective and efficient activities (Petticrew & Roberts, 2006). To make systematic reviews effective, there is a need for their involvement with different stakeholders, such as researchers, practitioners, and end-users, to ensure the relevance of their findings in real-life situations (Tricco et al., 2017).

However, systematic reviews have their own shortcomings despite their procedural stringency and feasibility. The issues mainly lie in the potential bias of the work, which could be due to publication bias, selection bias, and variability across the included studies (Higgins et al., 2019). For reaping the benefits of each of these values, the methodological aspects have to be ensured through comprehensive search strategies, rigorous quality assessment systems, and strong synthesis approaches (Page et al., 2021). Systematic reviews have been widely recognized as the most methodologically rigorous research synthesis in combining and analyzing data. However, their process also needs to be continually improved and changed to better negotiate the knowledge-generating process (Gough et al., 2017).

Systematic reviews are conducted with a particular methodology for data synthesis, and thus contribute to informing evidence-based decisions and behaviors (Moher et al., 2009). They make knowledge generation accurate, honest, and stakeholders are accountable for the results through considerations for transparency, accountability, and stakeholder engagement of the principles (Tricco et al., 2017). Systematic reviews have a great role in advancing academic debates and forming evidence-based policies on responding to practical questions given their exhaustive and comprehensive manner of work (Higgins et al., 2019).

2.2.1 Identifying Keywords

Identifying keywords in SLR methodologies leads to a profound and deep investigation. In SLRs, keywords help concretely capture the potent and appropriate queries in the search expression and are used to help acquire the research systematically. The researchers set meaningful definitions for key concepts and terms in the study, developing possible synonyms, variations, and associated expressions with the intention of assuring comprehensiveness and relevance in the process of searching. To perform this activity, one needs to team up with domain experts who have initially performed the experiments, while the first experiments on the query searches are advised so that these set keywords are further enhanced and validated. (Higgins & Green, 2011).

There is a need to use keywords in the conducting of Systematic Literature Review (SLR) procedures in accordance with the instructions on information retrieval and database search methods. To increase the level of scope in the search of literature related to the topic of their study, the researchers conduct an elaborate search of various bibliographical databases using the standardized search syntax and Boolean operators. The addition of the grey sources of the literature and manual searching tactics extend the literature search and improve its completeness. The search tactics need to be properly reported and described, and

this ensures that others can scrutinize and replicate them, thus enhancing the scientific reliability of the Systematic Literature Review (SLR).

Moreover, research is dynamic in nature, thereby necessitating the continuous development and refinement of keyword identification. Researchers must remain vigilant in watching the development of vocabulary, new ideas, and changes in disciplinary debates (Page et al., 2021). In the quest to remain current, searches should be refreshed and renewed at intervals (Gough et al., 2017). The iterative nature of keyword identification in systematic literature reviews gives primacy to flexibility and speed, so that the conclusions drawn from research are both admissible and timely (Petticrew & Roberts, 2006). The robustness in the systematic keyword identification in systematic literature reviews is methodologically invaluable, contributing to the knowledge base and useful in evidence-based decision-making in multiple disciplines (Tricco et al., 2017).

2.2.2 Synonyms

Synonyms are a critical aspect of research for the sole reason of allowing researchers to expand and augment their search strings, making the knowledge they can achieve wider and broader in scope (Booth et al., 2016). Appropriate evaluation and intellectual judgment are two things that need to be properly taken into consideration while applying synonyms, as they have various subtle nuances and touches (Hart, 2018). The use of synonyms in research widens the scope of investigation while introducing ambiguity and inconsistency into the information gathered, which will ultimately affect the accuracy and importance of the material acquired (Bailey, 2015). Accordingly, it is important to evaluate in detail the necessity and possible consequences of using synonyms while researching to optimize their effectiveness and minimize the potential negative impact (Gough et al., 2017).

An important obstacle related to synonyms is to efficiently manage the diverse linguistic usage across different themes, regions, and/or domains (Booth et al., 2016). It sometimes proves to be difficult to identify the synonyms that are applicable to a given topic and to select ones from the often numerous synonymous terms, abbreviations, or informal expressions (Hart, 2018). To increase both precision and inclusiveness, researchers must then dedicate necessary and thorough effort toward compiling an exhaustive compilation of synonyms—and do so using an array of varied perspectives and linguistic proficiency (Bailey, 2015).

Further, the choice and arrangement of synonyms are closely connected to broader considerations of research methodology and epistemology. The researcher's selection of synonyms reflects their grasp of the research problem, which consequently impacts the scope and limitations of the study (Gough et al., 2017). The implications of the selection of synonyms should be caefully thought through by the researchers – taking cognizance of the agricultural context, discipline expectations, and the expectations of the envisaged readers (Hart, 2018). Transparency and reflection are paramount in selecting synonyms. The researchers should offer a cogent rationale for their selection and note any biases or blind spots of which they may be aware (Bailey, 2015). The utilization of synonyms can be enhanced by researchers using a methodical and rigorous approach in identifying synonyms so that they conduct better literature searches and share knowledge more effectively within their disciplines (Booth et al., 2016).

2.2.3 Search String

Creating a search string for Systematic Literature Review (SLR) technique is key to a methodical gathering of important content that adheres to scientific rigor and allows for the reduction of bias. Researchers define a clear and specific topic of research interest from

the start, by determining the underlying concepts and variables of interest (Kitchenham & Charters, 2007). The first step lays the foundation for structuring the search query, by helping researchers see relevant keywords and synonyms that clearly embody the study topic (Booth et al., 2016). Keywords are then chosen so that terms, words, or language use, as well as academic traditions, are wide-ranging and all-encompassing, ensuring a full and comprehensive approach to the search technique (Cooper et al., 2018).

Table 2.0 Search String Method

No.	Search Engine	Search String	Result
i.	Web Of Science	TS= (("Dimple" OR "Dent" OR "Groove") AND ("Skin" OR "Layer" OR "Surface") AND ("Drag Reduction" OR "Reduce Friction") AND ("Computational Fluid Dynamics" OR " CFD Simulation" OR " Numerical Fluid Dynamics" OR "Fluid Flow Analysis"))	40
ii.	Scopus	ALL ("Dimple" OR "Dent" OR "Groove") AND ("Skin" OR "Layer" OR "Surface") AND ("Drag Reduction" OR "Reduce Friction") AND ("Computational Fluid Dynamics" OR " CFD Simulation" OR " Numerical Fluid Dynamics" OR "Fluid Flow Analysis")	305

In this regard, this search string is intended to find out articles that discuss the combined influence of side dimples in Ahmed body skin friction and drag reduction using CFD analysis. This is a well-thought-out strategy that will ensure that a wide range of relevant literature is included in the review.

They purposely use Boolean operators and search syntax to build up a chain of definite and logical connections that focus and narrow down the literature search. Researchers utilize the required operators, namely \"AND,\" \"OR,\" \"NOT,\" to specify the needed intersections or exclude them in the found text. The researcher may enhance the search string sensitivity and specificity by enhancing the total number of factors, such as

various word endings, spellings, or typographical errors, related to the use of truncation, wildcards, and proximity operators.

Keeping the above elements in mind, refinements to the search string will be made through pilot testing and discussions with subject matter experts in order to ensure that the search strategy adequately captures the width and depth of the landscape of the relevant literature necessary to meet the research issue. Clear documentation and reporting of the search string are essential for performing Systematic Literature Reviews, whereby practical insights across different research domains can be integrated through evidence development.

2.2.4   PRISMA Model

The PRISMA model is a structured guideline that seeks to enhance the transparency and quality of systematic reviews and meta-analyses (Moher et al., 2009). This comprehensive model contains a checklist, as well as a flow diagram developed to assist researchers at every step of the review model, ranging from formulating the original search strategy to integrating and presenting findings. The checklist consists of 27 items classified into areas such as title, abstract, introduction, methods, findings, and discussion. Under each such area, there are different items that need comprehensive information in the process of reporting the study and accelerating the assessment of the study's credibility and validity associated with the claims made.

The PRISMA methodology elaborates on the need for providing clear and detailed documentation of procedures and findings in systematic reviews. Researchers should accompany their reports with clear descriptions of search strategies, how studies were selected, data abstraction methods, bias assessments, and other crucial features. Adherence

to the PRISMA statement can enhance the quality and transparency of systematic reviews so that the methods are clear for readers to see how these have influenced the result.

Additionally, the PRISMA flow diagram illustrates the flow of information at different stages of the review process and complements the checklist. It lists the total records identified, records screened, records assessed for eligibility, full-text articles assessed for eligibility, and the total number of studies included in the review. The reasons for exclusion at each stage are also highlighted. The diagram eases the tracking of research progress and comprehending the reasons for study inclusion and/or exclusion by giving a clear and reproducible overview of the review process. The paradigm of PRISMA through pathways the credibility and relevancy in findings of systematic review research, whereby transparency,comprehensiveness, and replicability are prioritized. The prior stakeholderswho benefit from this are not only the scientific community but also other stakeholders.
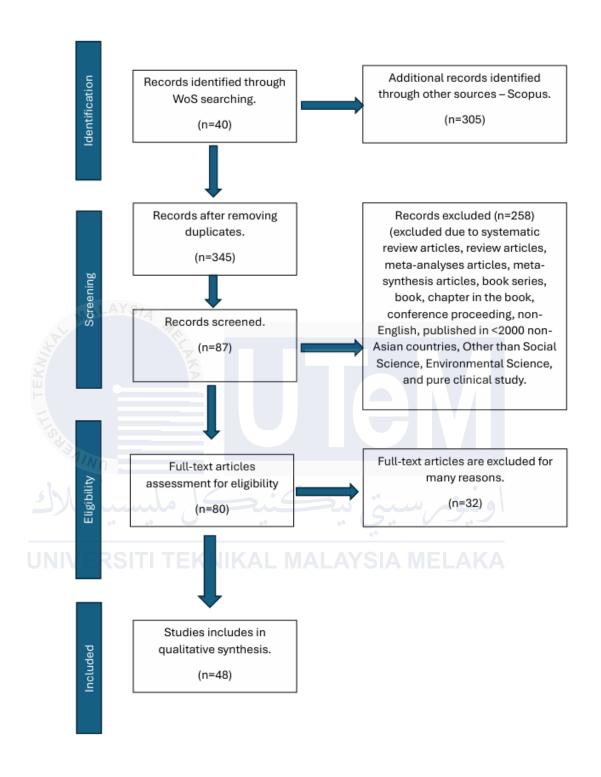
Figure 2.0 PRISMA Model

2.2.4.1 Identification

Limiting the number of papers screened to the WoS and Scopus webpages shows an approach that is focused on finding literature through two known databases with wide

coverage in various fields. Although WoS and Scopus are reliable sources of academic literature, relying on these databases alone may lead to the omission of relevant studies not found in their repositories. This limitation calls for the need to acknowledge the possibility of having gaps in coverage and search for other sources to ensure a comprehensive literature search.

Getting access to approximately 795 papers identified in the Web of Science indicates that a careful approach was applied from the identification phase of the systematic review procedure. However, the large number of articles that have been gathered requires scrutiny in order to determine the relevance and quality of the studies contained. Clear documentation of the search strategy, which will include the keywords and search terms, and inclusion or exclusion of criteria, is necessary for evaluation in terms of both search and procedural thoroughness and replicability. In a systematic review, rigorous screening and reaching inter-rater reliability in the study selection process will improve reliability and validity by ensuring that only high-quality and pertinent publications are admitted.

By using the WoS and Scopus websites to look for research material, one is able to systematically access scholarly works. Nevertheless, one needs to understand the shortcomings of exclusively relying on the databases. Proper documentation of the search strategy and systematic screening methods will ensure the systematic review is exhaustive and valid, thereby boosting the credibility and dependability of the results.

2.2.4.2 Screening

Screening within the PRISMA model is important to identify relevant quality journal articles while avoiding duplicate and irrelevant studies. Having an abundance of papers from both the WoS and Scopus at the outset indicates a good search of the literature; however, it

16

also creates problems with the number of records being pulled down. Finding duplicate information among multiple sources is a common problem in systematic reviews; correct deduplication processes are hence essential to arrange the information in an organized manner and to avoid redundancy. Deduplication reduces entries; however, processes should be documented and differences between the two datasets mentioned to ensure transparency.

Further refining of the dataset involves the application of rigorous inclusion and exclusion criteria to remove papers that are not aligned with the research objectives. The review focuses on relevant content, excluding review articles, meta-analyses, meta-synthesis articles, book series, conference papers, non-English papers, and studies emanating from non-Asian countries and those papers that are neither social sciences, environmental sciences, nor pure clinical in nature. The application of these criteria requires careful thought to ensure that useful studies are not inadvertently excluded. Transparent disclosure of the screening criteria and the rationale for any exclusion will be an important factor in enhancing reproducibility and dependability within the systematic review.

The final dataset for this systematic review comprises 87 journal article papers, identified through screening techniques to select appropriately high-quality and relevant studies. Although subjectivity in the screening may warrant several amendments, the review results' reliability and strength are improved by following rules and transparently reporting all processes. In so doing, researchers bring about transparency and scientific rigor in the PRISMA approach through well-documented processes of the screening exercise coupled with justification for each in and out decision.

2.2.4.3 Eligibility

When conducting an extensive systematic review, ensuring that the papers adhere to the PRISMA requirements is necessary. Filtering and selecting specific publications according to research goals and inclusion criteria used in eligibility assessment consist of applying evidence-based criteria to enhance the filtering. The key effort behind selecting articles is filtering by selecting those articles whose titles contain an exact approach toward the research subject. Since they rely upon the title for eligibility assessment, they can migrate important studies, offering valuable insights but do not specify the focused part of study within the title. Even though the title serves as a good beginning, comprehensive eligibility assessment may also be required through abstracts, keywords, and full-text analysis, hence ensuring a full and adequate procurement of as much relevant research information as necessary.

The inclusion of 32 journal articles without empirical data and relevance to both MAS and OSA emphasizes the need to ensure that the studies selected are within the scope and aims of the systematic review. The elimination of documents that do not contain empirical data focuses the review on that research which offers new findings proven with empirical data, thus enhancing the credibility and validity of a review. The selected papers would directly help build the information in the selected subsectors, helping present a focused and coherent overview of the literature. It is important to note that biases that limit the plenitude of points of view in the review are rooted in the exclusion of documents based on such criteria. The systematic review should transparently state the criteria for study eligibility and reasons for exclusion to ensure the review is transparent and reproducible.

The eligibility assessment under the PRISMA model calls for a judicious balance between specificity and inclusiveness, selecting those articles that closely relate to the research issue in terms of methodological rigor and relevance to the review objectives.

Highlighting the main subject matter, close adherence to specific themes such as MAS and OSA increases the cohesiveness and relevancy of the papers selected; there is a need to be cautious about biases emanating from exclusion criteria and thus to prioritize transparency. Researchers increase the trustworthiness and dependability of review results by clearly stating the eligibility criteria and reasons for the exclusion of each item, hence maintaining transparency and scientific rigor in the systematic review process.

2.2.4.4 Included

For the systematic review, forty-eight articles were identified using the PRISMA guideline and mark the strength of the choice of research for qualitative analysis. It must be given consideration about the value and relevance of the different journal articles with respect to the objectives of the study and synthesis of evidence. Estimations about the methodological credibility, conceptual clarity, and analytic depth of the chosen research are undertaken regarding the importance of the results, the suitability of the methods used, and the adequacy of the analysis. Improved reproducibility and reliability are the products of explicit disclosure of the study inclusion criteria and the basis for such choice.

The literature review to be conducted on the chosen journal articles is about understanding the common themes, differences, or gaps in it. Researchers can synthesize the findings of the selected studies by reviewing the key features or topics covered and by studying the relevant research themes. It is important to critically notice the included bias and shortcomings in the selected research and proclaim the method of conducting the synthesis and findings very clearly. Systematic reviewers maintain transparency and scientific rigor with the help of strict synthesis techniques and explicit reporting

requirements in the systematic review process to boost the credibility and reliability of the review findings.
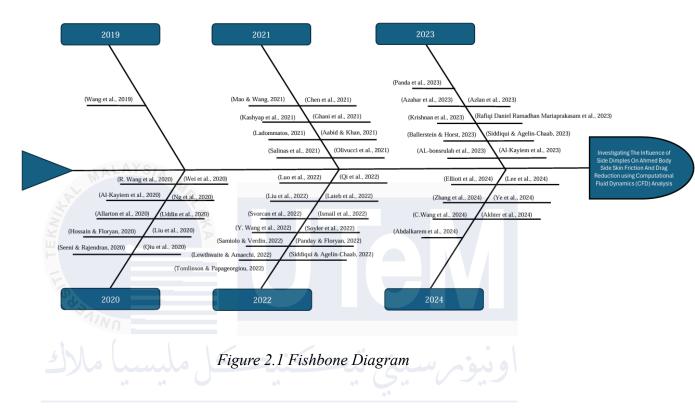
2.2.5   Background Study

To investigate cognitive landscapes in an elaborate manner, it is important to understand the elements and factors of academic research and knowledge creation. The Fishbone Diagram is an analytical instrument that may help structure, classify and analyze the available scholarly material that is relevant to particular topics. It provides an approach to in-depth analysis of complex topics by identifying main themes, points of weaknesses of the subject and brings attention to new directions in research. It shows how the ideas, theories, and techniques discussed in academia can be interconnected.

This branch-categorizes journal articles, research papers, or scholarly publications based on certain classification. This approach will call this technique since the appearance of this part of the paper is based on the anatomical structure of a fishbone. The use of the branches in the fishbone diagram shows the different categories or dimensions of the study, e.g. publication year, research topic, theoretical framework, methodological approach, or geographic region, from which a thorough review of literature can be conducted.

The Fishbone Diagram method entails the classification and analysis of journal papers of different years so as to describe the scope of academic inquiry into a topic of interest. The aim is to highlight historical patterns, thematic trends, and shifts in knowledge that have influenced the shaping of scholarly debates. Analyze literature, identify research gaps, evaluate the quality of existing scholarship, as well as obtain insights to inform our research agenda and improve our understanding of the issue.

The effort made is to better comprehend the topic and provide useful contributions to academic debate by conducting a sound and intelligent review of the available information.

Figure 2.1 Fishbone Diagram

The following will demonstrate the importance of findings in journal publications from the years 2019 to present to our study focus. Studying literature from previous years will enable an understanding of the quality of the standard topics within academic discourse, research methodologies and theoretical approaches that are offered. The analysis will provide information on the detection of links, inconsistencies, and new ideas in the literature, drawing attention to research that will be based upon the research aims of the one being prepared. As well as it will provide valuable insight into the progress and direction of academic research in this discipline to enrich the context and theoretical foundation of our research effort:

i) 2019

This numerical simulation studied how grooves on a superhydrophobic surface reduced drag. Scientists used CFD to research the way groove microstructure influences droplet behavior when hitting the surface. Simulation results presented three main findings: droplets distribute along grooves instead of perpendicularly, small groove-directed spheres at the droplet's edge and retraction droplets are slender and cross shaped.

The grooves on the superhydrophobic surface were researched to analyze drag reduction near the wall in a two-dimensional microchannel. Air trapped in grooves generates low-velocity vortices, rolling liquids on the air surface. This rolling causes velocity slip at the gas-liquid interface, resulting in drag reduction. Drag reduction was significant with laminar flow conditions from the superhydrophobic surface, but turbulent flow conditions exhibited less and even an increase in wall flow resistance.

ii) 2020

This collection presents a variety of aerodynamic studies aiming at efficiency, safety, performance in a variety of fields. (Hossain & Floryan, 2020) focused on novel techniques like 'golf-ball dimpling' at the aerofoils to decrease flow detachment using surface texturing like dimple textures to control friction and wear in sliding contacts. (Allarton et al., 2020) looked at practical issues facing the transportation industry. This includes the study of tire noise and friction in passenger car tires and an investigation into the low-tension piston rings and low-viscosity fluids in internal combustion engines. (R. Wang et al., 2020) aim to improve aerodynamic performance and decrease energy consumption. They also ensure durability in mechanical systems, which will correspond to advancements in fields, such as civil engineering, automotive, and the aerospace industry.

iii) 2021

The topics that follow present diverse spectrum of aerodynamic studies relevant in enhancing various aspects of engineering applications. The work is researching dimple-structured surfaces that enhance automobile piston component performance in terms of low frictions and increased load-carrying capacity. Another research is on the topic of "Laminar channel flows: pressure losses in presence of wall transpiration" to enhance the comprehension of flow stability. (Kashyap et al., 2021) research the techniques to reduce drag around cylinders by active and passive control approaches and striving for effective modification in the flow properties.The study has identified improving energy transfer techniques in the optimization of turbulence and in regulating heat conduction. Explicit studies have been done considering the influence of riblets. Research has also been carried out on minimizing vortex-induced vibrations of spheres, precise monitoring of flow characteristics close to blunt bodies, and quantifying the impact of disc motion over the turbulent channel flow to reduce drag. Ongoing research is the aerodynamic resistance of air rifle pellets, managing base pressures in high-speed fluid dynamics, and reducing aerodynamic drag on automobile surfaces. There is ongoing research about how vapor can be used to reduce the drag on textured surfaces at high temperatures in attempts to understand the relationship between surface roughness and temperature for the enhanced performance.

iv) 2022

The research studied numerous aspects of aerodynamics applied across a range of industries to improve performance, safety, and efficiency in vehicles. (Svorcan et al., 2022) researched surface vibration's effect on propulsion and resistance in parallel plate flow. (Soyler et al., 2022) research project is focused on reducing the drag of

aerodynamics at high speeds when wind resistance is high by utilizing vortex generators. An additional ongoing research project seeks to lower the forces inflicted by a landslide on pipelines buried in the ground parallel to studying the aerodynamic forces on long-distance buses to improve their stability. Fuel efficiency can be studied from the point of view of aerodynamics by examining vehicle shapes, including the Ahmed body at various incline angles. (Lateb et al., 2022) researched techniques such as streamwise grooves to find methods for enhancing flow properties and exploring the use of flow control devices in aeronautic and power applications. The use of superhydrophobic surfaces in channels can overcome the hindrance of smooth walls and enhance channel flows, such as for improving fuel efficiency. Improved novel heat exchanger channel design is being tested to allow vehicle propulsion to shift from Nature to electric power. (Tomlinson & Papageorgiou, 2022) studied the influence of lateral skirt devices on semi-trailer vehicle drag and flow structure, also researching a modified geometry to reduce drag by emulating the winglets used on aircraft wings. Scientific research is being conducted on cylinder and groove wakeregionsin amphibious vehicles to enhance the sailing speed and reduce fuel consumption. A bionicinto tire groove can enhance tire hydroplaning, and v-grooves reduce drag on cylinder surfaces.(Qi et al., 2022)

v) 2023

The research that is offered ranges widely in the content area of aerodynamics and fluid dynamics. Thus, (Azahar et al., 2023) examines the conversion of wind kinetic energy that varies through a converging region into mechanical energy, with concern for the presence of solar radiation. Another considers the impact that the presence of dimples has on the aerodynamic performance of wind turbines, while still another creates a complex model to describe the flow of fluids at low velocities and how that

flow impacts a system of channels with a patterned wall. Additionally, (Al-Kayiem et al., 2023) there is the consideration of improving the aerodynamic performance of these wind turbines and reducing the drag on vehicles through novel patterns of dimples. Also, the consideration of the improved energy efficiency from Savonius wind turbines and decreased vehicular drag as a result of decreasing pressure via the use of specialized patterns of dimples. Another is building an understanding of axi-symmetric body drag-reduction techniques and the motorsport induced to reduce vehicle drag while creating a better understanding of drag reducing technology on vehicles.

vi) 2024

The above studies represent a wide range of aerodynamics and fluid dynamics studies. One such examination investigates the lift and drag features of rotating spheres with rough elements, including spin rate and the number of roughness elements. Another investigates the stability of the laminar channel flow formations. Biomimetics and computational fluid dynamic research combine to enhance underwater vehicle design features. (Akhter et al., 2024) examination of axial corrugations on the cylindrical annuli to reduce drag and inclined slot features on air foils of wind turbines. (Abdalkarem et al., 2024) efforts to increase aerodynamic features by adding wedge flaps to the air foil. Efforts are made to increase qualities via adding wedge flaps to the air foil, with new riblet surface designs, while other efforts study the effect of the depth of the water impact on Air Layer Technology for drag reduction, in a pure sense, on shallow-depth water navigational vessels.

2.2.5.1 Modelling

Selection of a modeling strategy in research on fluid dynamics is very significant, considering the fact that it directly affects the level of understanding, ranging from detailed examination to the likelihood of conclusions at the study distance. Different modeling tools exist in current research, which can be used to study fluid flow phenomena, each with its variability and constraints. CFD simulation is a useful tool for fluid dynamists due to its diversity and capability of numerically solving complex fluid flow equations. CFD simulations support the analysis of complex fluid dynamics phenomena, such as turbulence flows, coupling multiphase interactions, and heat transport mechanisms, through digital modeling of physical systems into computer domains. CFD is applied in diverse fields, including aerospace engineering and biomedical sciences, for examining design alternatives, process optimization, and performance projection.

Numerical simulations, experiments, and theoretical research contribute to studies included within the scope of fluid dynamics, appealing with different views and opportunities for validation. Experimental research provides an opportunity to directly observe and measure fluid flow phenomena in controlled laboratory conditions, enabling empirical data for the validation and fine-tuning of computational models. Experimental research allows the study of real-world complexity and confirmation of theoretical assumptions, thereby boosting the dependability and applicability of theoretical models. Theoretical research is very valuable in the explanation of basic principles regulating fluid behavior and elaborating mathematical formulations describing fluid flow phenomena. The use of mathematical models and analytical tools to dissect the underlying mechanisms and predict system responses allows researchers to guide experimental insight.

Besides traditional modeling techniques, these improved methodologies, such as Design of Experiment (DOE) and Reduced-Order Modelling (ROM), offer new possibilities

26

to increase the level of fluid dynamics research. The Design of Experiments allows for the systematic exploration of a parameter space and the establishment of optimal conditions for several elements simultaneously changed and investigated to study their effects on the system's response. Through statistical approaches and response surface methodology, researchers can efficiently explore complex design spaces, optimize experimental techniques, and expose hidden correlations between the input factors and system performance indicators. Reduced-Order Modelling methodologies enable fast, computationally efficient approximations of complex fluid dynamic systems by reducing their essential dynamics to lower-order variables. Modal analysis and Eigensystem Realization Algorithm enable the simplification of intricate fluid flow problems for quick parametric explorations, sensitivity evaluations, and design of real-time control schemes.

Sensitivity analysis helps to understand the relationships that lie between the input parameters and the system's responses—most importantly, in the optimization and control problems. Through the measurement of the responsiveness of system outputs or outcomes to the changes in input variables, researchers can determine those parameters that are sensitive and the optimal operating conditions to design control systems for the elimination of drag and turbulence. Sensitivity analysis methodologies have been applied to various research issues such as automobile engineering and environmental fluid mechanics penetration for the improvement of decision-making support and, hence, performing effective, robust designs fostering innovation.

Recent methodologies and ideas such as wall transpiration analysis clearly have brought about the innovation and integrative aspects of contemporary fluid dynamics research. With the wall transpiration, researchers can regulate the characteristics of the fluid flow, modify the pressure distribution, and enhance energy transfer processes of engineering applications. By the theoretical analysis of the mechanics controlling wall transpiration

effects, numerical simulations, and experimental validation, researchers can identify and document the fundamental mechanisms and use them for system improvement and efficiency.

In fluid dynamics, the selection of modelling tools depends on the purpose of the study, the level of complexities in mimicking the fluid flow phenomena under investigations, and the ability, resources, and experience available. Through strategic integration of computational and experimental research in a complementary way with theoretical analysis and innovative research, efficient ways can be found for understanding the behavior of fluids to increase scientific knowledge and develop new solutions to complex engineering problems.

2.2.5.2 Methodology

Table 2.1 Methodology Used in Previous Studies

No.	Author	Turbulance Model	Input	Output
1.	(Ismail et al., 2022)	k-e turbulence model	CFD simulations of sedan car models with/without vortex generators.	Improved aerodynamic performance and reduced drag.
2.	(Ng et al., 2020)	Large Eddy Simulation	Impact of shallow dimples on turbulent drag in channel flow.	Reduction in turbulent drag and improved flow characteristics.
3.	(Liu et al., 2022)	RANS	Grooves on amphibious vehicles.	Enhanced hydrodynamic performance and reduced drag.

4.	(Salinas et al., 2021)	RANS	Flow near radars.	Improved radar accuracy and reduced aerodynamic interference.
5.	(Olivucci et al., 2021)	Large Eddy Simulation	Effects of rotating discs on turbulent channel flow.	Significant drag reduction and enhanced flow stability.
6.	(Ladommatos, 2021)	RANS	Surface pressure and drag analysis of pellets with varying geometries.	Optimized pellet design for reduced drag and improved flight stability.
7.	(Azahar et al., 2023)	k-ε turbulence model	Solid sphere at different Reynolds numbers.	Insights into drag characteristics at various flow conditions.
8.	(Ismail et al., 2022)	k-e turbulence model	CFD simulations of sedan car models with/without vortex generators.	Improved aerodynamic performance and reduced drag.
9.	(Siddiqui & Agelin-Chaab, 2022)	SST κ-ω turbulence model	3D wake dynamics of a slanted Ahmed body using IDDES.	Detailed wake structure analysis and drag reduction insights.
10	(Samiolo & Verdin, 2022)	RANS	Finless heat exchanger layout for electric vehicles.	Enhanced heat transfer efficiency and

				reduced aerodynamic drag.
11	(Qi et al., 2022)	RANS	Effects of V-groove depths on various aerodynamic factors.	Optimization of V-groove depths for improved aerodynamic performance.
12	(Azlan et al., 2023)	k-ω SST turbulence model	CFD simulations of rotor forces.	Improved rotor efficiency and reduced aerodynamic drag.
13	(Al-Kayiem et al., 2023)	k-ε turbulence model	Drag reduction in pipe flow.	Significant drag reduction and improved flow characteristics.
14	(Siddiqui & Agelin-Chaab, 2023)	SST K-Omega turbulence model	Numerical simulations using FLUENT solver.	Detailed flow and drag analysis for various geometries.
15	(Al-Kayiem et al., 2020)	Large Eddy Simulation	Near-wall turbulent flow over streamlined riblet-structured surface.	Reduction in drag and improved flow characteristics.
16	(Panda et al., 2023)	Reynolds Stress Modelling (RSM)	Shallow dimples' impact on drag reduction on axisymmetric bodies	Effective drag reduction and improved aerodynamic performance.

Present-day Computational Fluid Dynamics (CFD) research adopts a number of turbulence models to capture the intricacy of the turbulent flows realistically. This paper analyzes the methods used in prominent research that adopts several turbulence models, providing unambiguous indications of their individual applications and the procedures implemented to achieve valid and reliable results.

2.2.5.2a K-epsilon turbulence model

The k-ε turbulence model is often selected due to its optimal trade-off between accuracy and computational speed:

i. (Ismail et al., 2022) Using Computational Fluid Dynamics (CFD), the investigators in this study proceeded to investigate the aerodynamic features of sedan automobile models with and without the aid of vestigial vortex generators. The k-ε model was used classic, with the model being applied to study the effects of the generators, in terms of drag reduction. For ideal real-life driving conditions, the original mesh was used together with the corresponding boundary conditions.

ii. (Azahar et al., 2023) on the flow behaviour around a rigid sphere for different Reynolds numbers; a mix of empirical and computational fluid dynamics (CFD) simulation was conducted. In the process, mesh refinement was done in some specific regions to capture the boundary layer effects adequately.

iii. (Al-Kayiem et al., 2023) addressed a study specifically to reduce drag in pipe flow utilizing the k-ε model. They used fine mesh especially for the regions near the pipe walls to have the characteristics of the flow determined well. Also, they conducted parametric analyses on various pipe geometries along with numerous flow conditions.

2.2.5.2b Large Eddy Simulation (LES)

Large Eddy Simulation (LES) is a computational fluid dynamics technique that resolves the larger-scale eddies and models the smaller-scale eddies in flow turbulence. The reason LES is chosen over the others is based on its ability to represent large, scale-wise, turbulent formations, accurately:

i. (Ng et al., 2020) has analyzed the flow-induced behavior of shallow dimples in a study concerning the influence of such geometrical features on the behavior of turbulent drag in channel flow through Large Eddy Simulation (LES). The study was performed on a very fine quality mesh aiming to render with precision both the flow patterns within and around the dimples. The authors verified the previous by comparing their computer simulation results with independent experiments.

ii. (Olivucci et al., 2021) The purpose of this paper was to determine the effects rotating discs cause to the turbulent channel flow through Large Eddy Simulation (LES). A detailed mesh containing the rotating discs was used to gather the flow patterns in a detailed manner by using a time-dependent analysis method to understand the characteristics of the transient flow.

2.2.5.2c Reynolds-Averaged Navier-Stokes (RANS) equations

The RANS model has been widely used in engineering practices because of its ease of use and lower computational cost.

i. Research conducted by (Liu et al., 2022) dealt with the geometry of the grooves on amphibious vehicles at a refined level using the RANS model. The simulations performed were in the steady-state condition involving all groove regions with substantial mesh detailing to correctly predict the nature of the flow.

ii.   According to (Salinas et al., 2021) , flow patterns near radars were analysed using the RANS model of the Navier-Stokes equations. In this study, the 3D representation of the radar system was developed, and the refined mesh was used to analyse the flow dynamics around minute characteristics.

iii.   (Samiolo & Verdin, 2022) used RANS for optimization of a finless configuration on a heat exchanger for electric vehicles. They carried out a series of parametric studies for different geometrical configurations and used a refined grid to accurately capture the thermal and flow features.

iv.   (Qi et al., 2022) studied the effect of V-groove depth profile on various aerodynamic parameters using the RANS approach. This included carrying out careful mesh refinements in the vicinity of the grooves and validating the outcome with experiments throughout to ensure accurate predictions were obtained.

v.   (Ladommatos, 2021) performed a study to analyze surface pressure and drag of cylindrical pellets with different geometries by RANS. In the study, a structured mesh was used with increased resolution toward the pellet surfaces to capture the boundary layer effects accurately. In addition, many simulations were run on numerous geometrical configurations.

2.2.5.2d k-ω turbulence model for the SST (Shear Stress Transport) equation.

The SST k-ω model is favoured because to its high accuracy in forecasting flow separation and adverse pressure gradients.

i.   With all this geometry a numerical simulation was carried out regarding the three-dimensional wake dynamics of an inclined Ahmed body performed by (Siddiqui & Agelin-Chaab, 2022) using IDDES with the SST k-ω model. Several types of meshes

33

were used, including those with refinements in the area behind the object, to better capture the complex flow features. They then performed simulations that accounted for changes over time to study the unpredictable behavior of the wake.

ii. (Azlan et al., 2023) carried out computational fluid dynamics (CFD) simulations to examine the forces on a rotor. They used the SST k-ω model in their computation. The researchers had a high-order mesh to consider the rotor blades and study the complex solution topology they obtained. They later performed parametric experiments to evaluate the effects of diverse operational conditions.

2.2.5.2e Reynolds Stress Modelling (RSM)

Reynolds Stress Modelling (RSM) is therefore a complete method where, in the form of supplementary transport equations, the Reynolds stresses are explicitly solved.

i. Using Response Surface Methodology (RSM), (Panda et al., 2023) carried out research on shallow dimples' decreasing drag effects on axisymmetric bodies. They used an intensive well-made mesh around the dimples to account for the anisotropic effects of turbulence. They further validated their simulation quality by putting it in line with experimental data.

Every turbulence model has its own advantages and limitations that determine applicability among various CFD studies. The k-ε model provides a good compromise between accuracy and calculation time, but the ability to treat complex flow features in a solution could be problematic. The LES method is highly accurate but computationally expensive. RANS models are widely used in industry because they are practical, while the SST k-ω model is quite effective when the flow separates. Using the RSM method, the turbulence structure is well analyzed, but at a cost of significant additional computational time.

The methods studied highlight the different applications of numerous turbulence models in computational fluid dynamics (CFD). Choosing a model depends on the requirements of the research in terms of accuracy, computational power, and the nature of the fluid dynamics being studied.

2.2.5.3 Outcomes

In the literature, studies on the application of numerous turbulence models with Computational Fluid Dynamics to simulate complex dynamics related to turbulent flows are widespread. This review discusses the relevance of a number of turbulence models that have been applied in significant research results and are relevant to the utility and efficiency of such a model.

The k-ε turbulence model is commonly used because of its ability to balance computational efficiency and accuracy effectively. (Ismail et al., 2022) used the k-ε model to assess the aerodynamic effectiveness of the sedan vehicles, both with vortex generators and without them. The study showed a massive reduction in drag due to the addition of vortex generators, giving tremendous weightage to the effectiveness of the model in practical automotive application. (Azahar et al., 2023) used the k-ε model to study the fluid dynamics around a rigid sphere encompassing different sets of Reynolds values. Their focus was on showing the ability of the model to predict flow features correctly, along with boundary layer phenomena, hence proving the efficacy of mesh refinement in improving the accuracy (Al-Kayiem et al., 2023) performed a study optimized for drag reduction in pipe flow.

They were able to come up with this through the k-ε model and the implementation of a finer meshing method near the walls of the pipe. This approach was thus a definite

exploration of fluid flow and the actual identification of optimal conditions in drag reduction and showed the model fit enough for fluid flow investigation in engineering systems.

The selection of LES is spurred from its extraordinary preciseness in expressing large-scale turbulent structures, even if it has very high processing requirements. In their research work, (Ng et al., 2020) employed Large Eddy Simulation to investigate the influence of shallow dimples on turbulent drag in channel flow.

The meticulous arrangement and fusion of the complex structure with real-world data confirmed LES's capability to represent flow patterns effectively and reduce drag. This proves the LES's improved effectiveness in circumstances mainly requiring an accurate representation of turbulence.

(Olivucci et al., 2021) investigated the impact of rotating discs on turbulent channel flow through the use of Large Eddy Simulation (LES). The study that was carried out with the consideration of time passage and included a detailed meshing procedure is giving valuable findings related to the development of flow features and a significant drop in drag force. This, therefore continues to persuade that Large Eddy Simulation (LES) is an important tool for the simulation of complex fluid dynamics.

Reynolds-Averaged Navier-Stokes (RANS) models are widely used in engineering as they are simple and do not require much computation. (Liu et al., 2022) employed the Reynolds-averaged Navier-Stokes (RANS) approach to study grooves on amphibious vehicles. Their study successfully captured perfect flow fluctuation by using a very detailed mesh. This shows the feasibility of applying RANS in engineering projects where relative computation is not affordable. Application of the Reynolds-averaged Navier-Stokes (RANS) approach was done by (Salinas et al., 2021) in the study of radar flow patterns.

They developed a detailed 3D radar model and employed a fine mesh to accurately predict the flow movement patterns (Samiolo & Verdin, 2022) significantly improved the design of a heat exchanger within an electric vehicle using a numerical tool called RANS. Their study confirmed the high predictive capability of RANS for determining the heat transfer and flow characteristics of the system, particularly when using an extensive grid. (Qi et al., 2022) confirmed the reliability of using RANS to model the aerodynamic effect of the V-groove depth. Their research involved careful meshing and experimental verification, producing accurate results.

(Ladommatos, 2021) indicated that RANS could perform effectively in the investigation of unpredictable variations in pressure fields and drag components on the pellets with multiple shapes, due to the structured meshing's capacity to properly represent the behavior of the boundary layer. The k-ω-sst turbulence model has shown high efficiency in the accurate prediction of flow separation, especially for unfavourable pressure gradients.

(Siddiqui & Agelin-Chaab, 2022) carried out a study on the wake dynamics of an inclined Ahmed body. They used IDDES with the SST k-ω model to access the three-dimensional properties of the wake. The extensive observations of the wake structure obtained from their simulations depicted reasonably good correspondence between the dynamic flow characteristics.

This investigation used the SST k-ω model implemented in computational fluid dynamics (CFD) by (Azlan et al., 2023) to study the forces acting on the rotor. The researchers used high-resolution meshing techniques to be able to analyse complex flow features and carried out parametric tests under different operating conditions. Detailed meshing trials have shown the outstanding effectiveness of the SST k-ω model in cases characterized by complex flow features, like separation and adverse pressure gradients.

The Reynolds Stress Modelling (RSM) method provides an integrated approach through the solution of additional transport equations for Reynolds stresses. of their work, (Panda et al., 2023). studied the effects of shallow dimples on drag reduction over axis-symmetric bodies through Response Surface Methodology (RSM). Their careful implementation and extensive testing confirmed the faithful representation of anisotropic turbulence effects and underlined the potential of RSM in capturing highly complex features of turbulence.

The literature review is based on the wide applications and benefits of different models of turbulence in CFD research works. The k-ε model is popular in its good level of computation efficiency and high goodness, especially in regard to matters dealing with automotive and pipe flow applications. LES has a high level of accuracy for complicated and large-eddy turbulence, but the required computation resources are very large.

RANS models are commonly employed in industry due to their applicability and lower calculation expenses. The SST k-ω model performs well in instances of flow separation, whereas the RSM yields a detailed description of the turbulence structure at a greater computational cost. An appropriate turbulence model can be selected based on the specific needs of the study, the level of accuracy required, the available computational resources, and the nature of the fluid dynamics features under investigation. The possibility of reliable and informative results from CFD simulations critically relies on the appropriate choice of the turbulence model according to the context of the problem addressed in the research.

2.3 Ahmed Body Model

The Ahmed Body is a critical automotive form that embodies a cart vehicle shape. Ahmed's work experimentally provides a clear representation and visualization of the airflow behavior around his body, which much mimics the primary flow behavior of a vehicle. Ahmed's body, although seemingly ordinary and unimpressive, gives us some critical information about the pivotal features relating to bodies within the automotive sector.(Siddiqui & Agelin-Chaab, 2023) The Ahmed body is a groundbreaking development in vehicle aerodynamics, transforming the comprehension and modeling of complex flow patterns around bluff bodies. The historical trajectory reveals a story of cleverness, cooperation, and the quest for dominance in aerodynamics.

2.3.1 Development of Ahmed Body Model

(Siddiqui & Agelin-Chaab, 2023) Furthermore, historically, over the 1970s during the fuel crisis, characterised by oil producing nations imposing embargoes, a heightened level of global concern signified that a possible fossil fuel shortage was imminent. This challenge spurred broad research efforts oriented towards driving down fuel consumption in various industries with a major focus on the aerospace industry. Concomitantly, aerodynamic research on ground vehicles evolved, with an objective towards shape optimisation for minimisation of aerodynamic costs. However, the aerodynamics of a real vehicle is very complex, and this made the use of simple, generic models an excellent tool for research and testing because it is very difficult to analyse experiments done on a full vehicle. This led to the development of many generic models of vehicles, amongst which is the Ahmed body, which received a lot of acclaim at its introduction by Ahmed and co-workers in 1984 (Ahmed et al., 1984). Due to the inherently atypical geometry of the Ahmed

body, it has become the focus of attention in many aerodynamic studies over the years and remains a versatile tool for investigations focusing on various aerodynamic phenomena and design changes.

2.3.2 Experimental Studies on Ahmed Body

Using the Ahmed Body model in computational simulations enables research into vehicle aerodynamics, giving both advantages and difficulties. The model represents the complex geometry of real automobile designs in a more straightforward manner and yet clearly has the key flow features reminding those of the flow around automobiles. However, the important simplifications of the numerical model of real automobiles require careful analysis of simulation conditions and computational domain configurations to ensure accurate results. The Ahmed model is a simple but yet effective configuration developed by (SR Ahmed et al. 1984) and is characterised by a circular front, rear slant angle that can be adjusted, and a cuboid joining the front and rear slant. The Ahmed model, while very simple, is generally representative of the basic characteristics of an automobile, allowing the changes in drag coefficient and wake separation phenomena to be closely analysed at different slant angles.
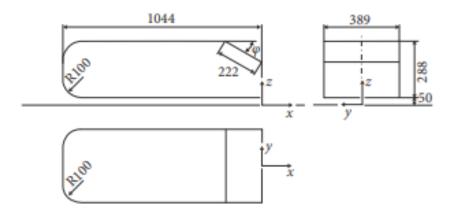

Figure 2.2 Ahmed Body Dimensions (Luo et al. et al., 2022)

40

Experimental studies were performed by (Luo et al., 2022) to analyze the aerodynamics of single, two, and three-vehicle 35° Ahmed groups. The researchers conducted validation of their numerical methods using a single Ahmed model by comparing the results of a two-vehicle group to empirical data. They found large differences in the drag coefficients: reductions were noted for the intermediate and rear vehicle of a three-vehicle convoy.

Higher mesh resolutions give more accurate results but at the higher cost of processing resources, as evidenced by means of comparing drag coefficients at different mesh densities. For the same reason, schemes such as Mesh 4 or Mesh 5 can typically be selected to present a compromise between the efficiency of the calculations and accuracy of the results while ensuring typical kinds of reliable simulations for the study of complex aerodynamics, including in platooning scenarios. (Luo et al., 2022)

2.4   Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) represents a powerful inter-disciplinary technology which combines the principles of fluid mechanics with recent groundbreaking achievements. Processing Fluid Dynamics (CFD) combine numerical simulations to approximate equations of fluid control, and use actual computer processing powers. The technology of CFD has long been used in various scientific branches for understanding complex fluid phenomena. (Luo et al., 2022) implemented Computational Fluid Dynamics (CFD) to study the aerodynamic characteristics of platoon vehicles with the Ahmed Body model.

Their work validated the precision of numerical approaches by comparing the computational fluid dynamics (CFD) results against empirical data. When the vehicles were

arranged in a platoon configuration, crucial drag coefficient differences were noted after performing this analysis. In the study conducted by (AL-bonsrulah et al., 2023) the flexibility associated with CFD was executed. In this work, they delivered a novel 3D model for analyzing solar radiation, built on the ANSYS Fluent software. This work shows that Computational Fluid Dynamics (CFD) can be applied in very many areas.

By using computational fluid dynamics (CFD), dynamic systems affected by natural occurrences have been analyzed. According to (Zhang et al., 2024), computational fluid dynamics (CFD) can be used to undertake examinations and obtain important inferences from live structures with a focus on developing the most remarkably efficient designs of underwater vehicles. Their research mainly focuses on improving propulsion, reducing drag, and eradicating noise. (Y. Wang et al., 2022) used computational fluid dynamics (CFD) to predict the aerodynamic forces that arise from buses. To demonstrate their predictions, they conducted wind tunnel experiments as well as used several drag reduction techniques, such as dimples, riblets, and fins.

In the research of (J. Wang et al., 2019) it is evident that CFD has a dramatic effect on advancing the understanding of complexed fluid dynamics. It targets the mobility of the droplet over bio-inspired surfaces with the help of advanced methodologies like the VOF method and the interface tracking method. Through such interdisciplinary approaches, it compiles information on bionics, interface science, and fluid dynamics for improved insights into drag reduction methods, making it easier to put the latter into practice.

2.4.1 Mesh Generating Technique

Generation of meshes is one of the most critical tasks in CFD simulations as it has a direct impact on the quality and effectiveness of the results obtained. Structured meshing is simple to employ and feasible in the case of simple geometry or where the regular grid pattern can accurately represent the flow characteristic. In their work, (Mohd Basri et al., 2020) state that a structured mesh is computationally efficient since it requires less memory and resources. However, it might be problematic for complex geometries or regions that require fine-detail attention. Such limitations may result in (and do at times) simulation errors and lead to direct interference in order to ensure the quality of the grid, which is a very time-consuming procedure.

Unstructured meshing allows for highly flexible and adaptive methods to be applied, providing the ability to better fit complex geometries and in turn allowing for better mesh resolution, recording more accurate flow phenomena. According to (Uddin et al., 2020) an unstructured mesh can tightly hold an irregular geometry, thereby enabling the exact representation of complex flow features, including boundary layers and vortex shedding. In the high-quality creation of unstructured meshes, large computational resources and time are necessarily used. It is therefore large to handle refinement and optimization techniques with great care so that the mesh does not give rise to certain problems like distortion.

The hybrid meshing approaches combine the good points of both the structured and unstructured methods to efficiently utilize computational resources and ensure precisions of solution accuracy. Hybrid techniques were reported by Chen et al. (2020) and Wang & Li (2017) in which structured meshes were used in the simple geometric regions and unstructured meshes were used in the complex parts. It provides a trade-off between efficiency and accuracy. However, rather conflictingly, Yang et al. (2021) and Kim & Park (2018) have stressed that the handling of the transition itself from very structured to very

43

unstructured seems somewhat intricate, making the mesh generation and preprocessing convoluted.

Each of these meshing approaches has its individual advantages and limitations. Selection of the preferred meshing technique has to be made based on the specific requirements of the flow scenario, for example, the complexity of the geometry, the desired level of accuracy in the solution, and the available computer resources. In this regard, a comprehensive assessment of the advantages, limitations, and areas of applications of structured, unstructured, and hybrid meshing techniques would be helpful for scientists so that accurate and reliable computational fluid dynamics simulations could be realized under a wide range of flow conditions.

2.4.2   Turbulence Modelling

Turbulence modelling is one of the most critical areas of computational fluid dynamics. The capability to represent faithfully the characteristics of turbulent flows is critical in any attempt to perform dependant simulations. Different turbulence models are used depending on the level of flow complexity and the computational resources available.

k-ε. It is one of the standard turbulence models used in practice due to its reasonable compromise between computational economy and accuracy. For example, (Ismail et al., 2022) conducted a study to determine the aerodynamics analysis of sedan cars with and without vortex generators. The results of their study showed a significant reduction in drag when using vortex generators, which explains the effectiveness of the model in real-world automotive applications. Similarly, (Azahar et al., 2023) conducted a study to evaluate the flow dynamics around a solid sphere at various Re using the same k-ε model. They observed that mesh refinement was effective in improving the solution's accuracy.

44

Large Eddy Simulation is chosen due to its accuracy in representing large-scale turbulent flows that are very computationally expensive. For example, (Ng et al., 2020) carried out a study on how shallow dimples affect turbulent drag in channel flow using Large Eddy Simulation. Their careful mesh and comparison with empirical data validated the capability of the model to represent flow patterns with reduced drag. Other studies include work done by (Olivucci et al., 2021) on the effect of rotating discs on turbulent channel flow, and their results granted elusive transient flow properties.

Reynolds-Averaged Navier-Stokes (RANS) models are extensively used in engineering because of their simplicity and lower computational resource requirements. (Liu et al., 2022) used the Reynolds-Averaged Navier-Stokes (RANS) technique to study the influence of grooves on amphibious vehicles. The study effectively put forth the variations in flow with high accuracy, owing to the employment of a carefully refined mesh. (Salinas et al., 2021) used RANS to study the flow patterns of radar, while designing a 3D model of the radar to mimic the actual flow physics.

(Samiolo & Verdin, 2022) performed an optimization analysis of a heat exchanger with no fins in electric vehicle layout using RANS. The study emphasized the accuracy of RANS in representing both thermal and flow characteristics. The SST k-ω model is outstanding in providing an accurate prediction when it comes to flow separation and adverse pressure gradients.

In their study, (Siddiqui & Agelin-Chaab, 2022) addressed an investigation of three-dimensional wake dynamics of an inclined Ahmed body. They used the IDDES and included the SST k-ω model to portray the model's accuracy in representing the time-dependent flow characteristics. (Azlan et al., 2023) use the SST k-ω model for their CFD to carry out the analysis of rotor forces.

The complexities of the flow pattern are addressed through maximum resolution in the meshing. Reynolds Stress Modelling (RSM) is a comprehensive approach through the solution of additional transport equations for Reynolds stresses. (Panda et al., 2023) evaluate the shallow dimples' impact on drag reduction on symmetrical bodies through RSM. They attain accurate portrayal of anisotropic turbulence effects by near refinement along with experimental validations.

2.5   Side Dimples on Aerodynamics

When advanced appropriately, surface texturing can be one of the critical techniques used in engineering applications to enhance material performance optimally. This improves the friction and wear resistance, increasing the load-carrying capacity of the different materials. (Ghani et al., 2021) provided the evidence on the efficiency of the dimpled textures in the systems of lubrication and reduced the friction and opposing wear conditions to a significant degree. (Krishnan et al., 2023) also provided an emphasis on the diversified use of dimple shapes in many areas such as aerodynamics, hydrodynamics, thermodynamics, counting to tribology.

Dimpled textures show significant promise in the applications regarding hydrodynamic lubrication. (Wei et al., 2020) performed CFD analysis to study the geometric characteristics associated with dimple textures. They have identified the importance of the dimple surface angle to form hydrodynamic pressure. Recently, (Ghani et al., 2021) used three-dimensional computational fluid dynamics (CFD) to study the performance of dimpled textured surfaces in effecting hydrodynamic lubrication. They used dynamic-assisted tooling turning (DATT) to generate different types of dimples and set the parameters of machining using Taguchi orthogonal arrays in an optimized manner.

Dimpled surfaces, such as those in golf balls, serve practical applications in aerodynamics. For example, it is proved by (Lewthwaite & Amaechi, 2022). that adding dimples on NACA 0017 aerofoils enhances lift effects while limiting drag to a minimal effect. Therefore, the current studies emphasize that with the use of dimpled surfaces, the aerodynamic efficiency could be optimized with the reduction of drag and an enhancement in flow characteristic.

2.5.1 Experimental Studies on Dimpled Surfaces

Computational Fluid Dynamic (CFD) simulations yield an excellent comprehension of the advanced fluid dynamics phenomena controlling dimpled surfaces (Ballerstein & Horst, 2023). Through CFD simulations, the pressure generation and load capacity under hydrodynamic lubrication can be considered through the solution of the Navier-Stokes equations, which contain convective inertia, as demonstrated by (Singh & Awasthi 2015).

(Wei et al. 2017). Dimple shapes like rectangles, circles, and semi-ellipses have also been observed in prior research based on three-dimensional CFD simulations. These studies found the high impact of Reynolds number and dimple dimensions on load-carrying capacity and pressure generation during hydrodynamic lubrication (Wei et al., 2017; Singh & Awasthi, 2015).

(Ghani et al., 2021) Three-dimensional Computational Fluid Dynamics (CFD) simulations were carried out with Ansys-CFX software, to analyze the behavior of hydrodynamic lubrication in dimpled textured surfaces. Dimples were fabricated using DATT's turning method while employing machining parameters optimized with the aid of Taguchi orthogonal arrays. Comparing dimpled and untextured surfaces enhanced

understanding of pressure distribution and load capacity that is critical for an improved tribological performance.

The numerical study was done by (Wei et al., 2020) using Computational Fluid Dynamics to investigate the effect of various dimple texture shapes on pressure accumulation. Their work focused on dimple shapes, depth, minimum film thickness, hole density, and now the factor of dimple surface angles. Therefore, this paper shows the effectiveness of dimpled surface textures with friction reduction through the result of pressure contours, pressure distribution, and estimated coefficient of friction.

Other research has indicated that from (Lewthwaite & Amaechi, 2022), the improved performance model of the cargo airplane improved its aerodynamics. Winglets alter the direction of flow of the airflow and provide for better blending of the wake on the wing with the aerofoil to produce a 16.31% minimize in wing pressure at an 82 m/s velocity. From the research, it makes the creation of small dents or indentations made on the aerofoils of NACA 0017, dimpling, is found to enhance the production of the lift effects at the cost of only a 6% increase in drag. The research experienced some difficulty validating the techniques, but they were able to use alternative techniques and made recommendations for future studies.

2.6   Skin Friction and Drag Reduction

(Olivucci et al., 2021) highlighted that skin-friction drag reduction in the turbulent flow of fluids is an important area of research for better fuel economy and to mitigate ecological impacts by significantly reducing fuel consumption and carbon emissions in various industries. Frictional drag thus offers development potential in the strategies for overcoming aerodynamic resistance in the effort toward fuel economy and the reduction of ecological impacts.

This is very effective in reducing passive drag, where external energy is not required, and the methods involve aeration of turbulence near the wall. (Rafiqi Daniel Ramadhan Mariaprakasam et al., 2023) had revealed research in classifying the strategies as passive and active; in passive, most of the strategies target the shape of the wall surface. Qiu et al., 2020 added that since passive techniques do not require an external power, in general, fluid interaction with complex surface try to keep the vehicle attached to the wall in one way or another without the active addition of normal stressors to the surface but have the potential to generate added friction. (Qiu et al., 2020)

(Elliott et al., 2024) presented a study for aerodynamic force on turbulent rotating spheres with prominent objects of roughness of substantial dimensions. The mean aerodynamic forces they were thrusted in the working of the lift and drag forces. It was observed that the increase in the number of irregularities increased the value of resistance and decreased the lift. Wang et al. (2024) highlighted this effect of roughness features on drag and lift and stated that higher quantities of roughness elements result in increased quantity of drag.

In automotive applications, the reduction of aerodynamic drag is vital for improved vehicle efficiency and the lowering of energy consumption. (Mao & Wang, 2021) supported the statement that flow management solutions to mitigate pressure drag are developed due to flow separation in the rear section of vehicles. According to the results of (Al-Kayiem et al., 2023), both types of methods, active and passive flow control, provide considerable drag reduction. However, it is of paramount importance that future studies develop standardized criteria to support data coherence.

2.6.1 Drag Reductions

The study by (Elliott et al., 2024) aimed to analyze the aerodynamic loading of rotating spheres who are fitted with large-scale roughness elements, taking consideration of lift and drag; the paper indicates that the drag is reliant on the wake dimension and the upstream separation point asymmetry. (C. Wang et al., 2024) indicate that the rise in the number of roughness features sees a considerable rise in drag. The roughness elements also transfer force to the surface thus higher drag, while an increase in roughness elements sees a downslope in lift.

The automobile industry has been keen on "the reduction in automotive aerodynamic drag, which can potentially make a direct influence on the overall vehicle performance increment and power consumption drop, therefore conserving energy in them" (Mao & Wang, 2021). First and foremost, the origination of drag is by pressure drag, which henceforth, is "generated when the pressure in the rear part of the body" need be lower compared to the one in the front part of the body. (Y. Wang et al., 2022) The flow separation in the rear region, as indicated by the wake, creates a zone of low pressure that contributes to the drag force. (Al-Kayiem et al., 2023)

The techniques that could be taken to solve this problem are control over the flow which can be active or passive: Active needs power input while passive do not. But the efficiency of those methods is not the same, and to truly generalize it, standard parametric values should be set to move forward in studying to reduce the inconsistency between the values from various studies and other experiments from around the world.(Al-Kayiem et al., 2020)

2.6.2 Active Drag Reduction Techniques

Active drag reduction methods use energy from the outside to vary the flow field in the surrounding of a solid object, which gives less aerodynamic drag.(Mohd Basri et al., 2020) These methods include the use of suction, blowing, or plasma actuators for changing the characteristics of the boundary layer and delaying the separation of flow. (Mao & Wang, 2021) explored active flow control methods to lower drag for automotive applications, demonstrating high prospects for these methods in vehicle aero improvement and fuel consumption reduction.

Also, (Al-Kayiem et al., 2020) have recently considered an approach to reducing drag that uses turbulent flow modifiers in pipe flow. They showed that it is possible to successfully modify the flow characteristics by implementing active techniques like the use of turbulence modifiers and bring drag down in virtually any fluid system. This highlighted the greater potential of active drag reduction techniques in improving performance for a range of applications from automotive to industrial fluid systems.

2.6.3 Passive Drag Reduction Technique

Passive drag reduction measures aim to reduce drag through design modifications in shape or surface characteristics of an object using materials without the input of additional energy. The passive strategies include changes in roughness of the surface, riblets, and dimples. Passive measures have the advantage that they do not demand any additional power. These can easily be integrated with existing design setups. (Chen et al., 2021)

(Krishnan et al., 2023) noted that the performance of passive techniques like dimpled surfaces in reducing drag and improving aerodynamic efficiency was equally effective. Their study revealed that some types of surface textures could influence flow properties significantly while at the same time reducing resistance in various applications. In the same

51

perspective, (Ghani et al., 2021) demonstrated that the application of dimpled textures was effective in promoting tribological performance by reducing both friction and wear.

In line with this fact, (Al-Kayiem et al., 2023) observed that in automotive aerodynamics, the use of passive flow management techniques can greatly reduce drag and improve vehicle performance. Such findings make it clear that passive approaches will have the potential for a significant aerodynamic benefit without complicated control or additional energy input.

2.6.4   Experimental Validation Methods

Validating experiments are essential, so one has to invent a way to test if these different drag reduction tactics actually work. This is because (Ye et al., 2024) claimed that the assessment of aerodynamic improvements in both wind tunnel tests and actual flight tests, supplemented with CFD modeling, yields critical information on the aerodynamic performance of the various designs. Each one of them has its own benefits and downsides.

Wind-tunnel testing allows for controlled conditions in measuring key aerodynamic features, although it is only partial in replicating real-life conditions. Flight testing, in turn, is the most realistic but is conducted at extremely high costs and time undertaking. Computational simulations provide an analysis tool in great detail but depend on precise modeling and validation using experimental data.

(Reneaux.,2004) and (Quadrio.,2011) pointed out that empirical data and thorough validation were necessary in ensuring reliable results for research in fluid mechanics. The recognition of limitations and possible error sources in validation experiments will enhance the credibility and accuracy of results in elucidating the phenomena of drag reduction as well as its application in practice.

2.7   Summary

The research work, "Investigating the Influence of Dimples on the Side of Ahmed Body Skin Friction Using Computational Fluid Dynamics (CFD) Analysis," aims at investigating the influence of the dimple on the skin friction of the Ahmed body using numerical techniques. The Ahmed body is a simple model used to simulate and understand the aerodynamics of vehicles.

CFD is an effective technology by which fluid mechanics ideas can be brought to bear with the developments in computer science to numerically simulate fluid control equations. The adaptability of CFD has been witnessed in different research works, including those related to the aerodynamics of platoon vehicles and submersible designs, which have highlighted its capacity to provide intricate knowledge about the complicated fluid dynamics phenomena (Luo et al., 2022) and (Zhang et al., 2024).

Turbulence modelling is of paramount importance in computational fluid dynamics to ensure proper representation of various turbulent flow attributes. In the past, different generations of turbulence models have been applied to explore different aspects of CFD, such as the k-ε model, Large Eddy Simulation (LES), and Reynolds-Averaged Navier-Stokes (RANS); the same is applied in the existing research work. For example, the k-ε model is used to evaluate the aerodynamic performance in automotive applications, while LES can uniquely provide high accuracy in simulating turbulent structures of large amplitude (Ismail et al., 2022; Ng et al., 2020).

The ability to properly predict and validate CFD in a numerical solution has made mesh-generating techniques important. When dealing with simple geometries, structured grids have computational efficiency, while unstructured grids offer a high degree of flexibility in dealing with complex shapes. Recent years have seen the usage of hybrid

meshing approaches to harness the benefits of both methods to effectively use computational resources while maintaining solution accuracy (Mohd Basri et al., 2020; Uddin et al., 2020).

The use of dimples to reduce passive drag has exhibited large potential in a myriad of applications, ranging from tribology to aerodynamics. These surfaces have shown the ability to reduce friction, increase the ability to carry weight, and enhance aerodynamic efficiency by leading to a change in flow characteristics of the surface over which they are applied (Ghani et al., 2021; Krishnan et al., 2023). Computational fluid dynamics (CFD) simulations have been useful in the understanding of both the effect of dimples on fluid flow and pressure distribution. The CFD simulations have provided insight into the usefulness or effectiveness of dimples in drag reduction (Wei et al., 2017).

The research associated with skin friction and drag reduction is the most important study in the field of fluid mechanics in terms of relation to fuel consumption and environmental sustainability. In order to reduce skin friction, both active and passive methods are used, and passive methods, including dimples, are a very direct and low-power-consumption approach. It has recently been proved in the literature that the use of passive drag reduction approaches greatly enhances the aerodynamics of a vehicle and reduces the drag very efficiently (Olivucci et al., 2021; Rafiqi et al. et al., 2023).

The study at hand focuses on the prevalence of dimples for skin friction on the Ahmed Body using CFD analysis. Increased pressure is given to the fact that the CFD process is efficient in simulating fluid dynamics; turbulence modelling is very important; mesh creation significantly affects the simulation; and finally, dimpled surfaces offer reduced drag and enhanced aerodynamic efficiency. This paper aims to build upon these findings by elaborating on how dimples on the side of the Ahmed Body can lead to a convincing reduction in skin friction. This would be achieved by making utmost use of the strengths of Computational Fluid Dynamics (CFD) to run accurate and reliable analysis.

CHAPTER 3

METHODOLOGY

3.1   Preliminaries

This chapter describes the methodology applied in conducting the study on the effects of the side dimples on the side skin friction and drag reduction of the Ahmed body, with the use of Computational Fluid Dynamics. The method involves two main components, of which are, first, a benchmarking process and, second, the analytical phase.

The first step is to create an accurate three-dimensional model of the Ahmed body within the CATIA V5 software. The model was exported and brought into ANSYS to perform a CFD analysis.

From the literature, accurate results can be obtained by developing high-quality meshes. (Uddin et al., 2020) used an unstructured mesh to represent the complicated geometry clearly: the mesh must adhere as much as possible to the sensitive surface of the Ahmed body. The user gives specific values of the boundary conditions and solver parameters to replicate the conditions of the aerodynamics application; below are the recommended approaches for the turbulence models by (Ismail et al., 2022).

A grid sensitivity analysis is carried out to validate the accuracy and reliability of the results. According to (Liu et al., 2022), this technique aims to assure sufficiency of the meshing resolution to the effective representation of certain interested flow features without overly consuming computer costs. They particularly focused on the research of grooves observed on amphibious vehicles.

55

In this, simulations are done on the baseline Ahmed body, and this is to set a reference point for the benchmark technique. The accuracy of such conclusions is established through comparison with established data, just as the validation methods adopted during studies on the aerodynamics of platoon vehicles by (Luo et al., 2022) First, the Ahmed body is modified to include lateral dimples. Again, the same set of steps, meaning creating a mesh and verifying its quality and setting boundary criteria, are repeated.

Next, CFD simulations are run to analyse the aerodynamic impact of the dimples. The obtained results are compared to the baseline measurement in terms of reducing air drag and enhancing the friendly aerodynamic performance. This procedure falls in the steps taken by, for example,(Ghani et al., 2021) where the CFD method was utilized to investigate the effect of surface texturing on tribological behaviour.

Thus, a detailed procedure to perform CFD analysis is provided in this chapter to ensure that each step is conducted in a systematic manner for coherent and fruitful results. This study, which has painstaking validation along with analytical approaches done, is an attempt to provide reliable insights into the aerodynamic benefits of side dimples on the Ahmed body.
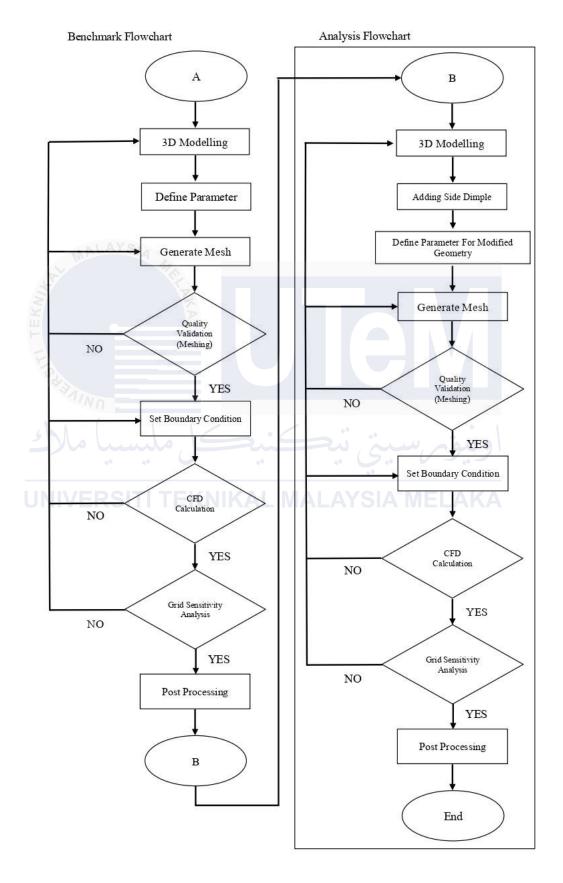
3.2 The Flow of Benchmark and Analysis

Figure 3.0 Benchmark and Analysis Flowchart

3.3   Pre-processing

Further in the section, the Computational Fluid Dynamics (CFD) analysis is extensively explained in how the simulation takes place and the fluid flow properties around the Ahmed body in both cases, with and without dimples on the sides. This process involves a number of very essential phases of work carried out, such as that of selecting appropriate models and creating the computational fluid dynamics (CFD) environment, employing methods that would guarantee precision in simulating fluid flow(Aabid & Khan, 2021).

3.3.1   Meshing

After finalizing the meshing and adjusting the boundary conditions, the CFD simulations began with the meshing process which is being performed using ANSYS Meshing in the Fluent module, which meshing using ANSYS Meshing is much more stable than other meshing software. This approach captures the flow features surrounding between the benchmark and modified model through advanced meshing techniques, including a structured and unstructured grid, the mesh refinement and also the boundary layer meshing. These steps are to ensure the representation of the Ahmed Body, benchmark and modified model, and provided reliable results for the aerodynamic analysis.

3.3.2   Modelling

Geometric modelling and mesh creation in Ahmed body applications are two critical aspects of CFD simulations as they shape the dependability and precision of the numerical results. This chapter describes the methods by which a high-quality computational model may be developed and constructed so that it reasonably represents the details of the Ahmed body continuum, in both the presences and in the absences of side dimples. In the first step, a new part is opened in CATIA V5; close attention is paid to developing the underlying

geometry of more realistic Ahmed body. Dimension values are established, according to standard Ahmed body criterion, which is guaranteed to result in precision and uniformity required for significant aerodynamic study.

Geometrical modelling begins by drawing a rough sketch of the Ahmed body's primary shape, comprising the front half, rear slanted half, and the lateral profiles. The drawn profiles are extended to develop the three-dimensional body, and fillets are added to refine the edges exactly as seen in realistic scenarios on aerodynamic surfaces. High-quality surfaces are developed using CATIA's surface modelling features, which closely represent the complex features of the Ahmed body. There is a need to check or ensure the surfaces that are created for gaps or jumps to be corrected. The gaps and jumps present in the surfaces will most likely affect the meshing to follow as explained below.

Side dimples are designed after establishing the base line of Ahmed body geometry. After fixing the sidewalls of the Ahmed body, the dimples of certain shapes and dimple proportions are placed on the sidewalls. Certain dimensions are fixed on the dimensions of the depth and shape of the dimple; these are most important variables that affect the aerodynamic analysis.

3.3.3 Boundary Conditions

Table 3.0 Boundary Conditions Setup

Boundary Conditions	Value
Model k-epsilon	2-Eqn
K-epsilon	Standard
Near-Wall Treatment	Standard Wall Functions
Cμ	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 [m^2]	0.05756999
Density [kg/m^3]	1.225
Length [m]	1.044
Temperature [K]	298.15
Velocity [m/s]	19.82
Viscosity [kg/(ms)]	1.837e^{-5}
Ratio of Specific Heats	1.4
Initialization Z Velocity [m/s]	-19.82
Turbulent Kinetic Energy [m^2/s^2]	1.473122
Turbulent Dissipation Rate [m^2/s^3]	1337.052

The CFD simulation will be performed with a standard k-epsilon turbulence model, one of the most well-known two-equation models for simulating turbulent fluid flow. The model contains transport equations for the turbulent kinetic energy (k) and its dissipation rate (ϵ), which gives a robust framework for the capture of flow behaviour. Standard Wall Functions are used to model the flow near walls, which is very important in the correct prediction of the dynamics within boundary layers. There are turbulence model constants, $C\mu$ =0.09, C_1−Epsilon = 1.44, and C_2−Epsilon = 1.92, that control the creation and absorption of the turbulent energy. Besides, the Prandtl numbers for the turbulent kinetic energy and dissipation rate are equal to 1 and 1.3, correspondingly.

The flow conditions in the simulation include the magnitude of a velocity of 19.82 m/s along the Z-axis, as per the projection direction. There is a specified minimum feature size in the computational domain, which is 0.001 m to provide an appropriate mesh resolution. The fluid physical properties consist of an area of 0.05757 m², a density of 1.225 kg/m³, and dynamic viscosity of 1.837×10 −5 kg/ms. The characteristic length is 1.044 m. In this simulation, the standard conditions were considered: the temperature was taken as 298.15 K with a specific heat ratio equal to 1.4 for air.

Initial conditions provide an initial velocity of -19.82 m/s in the Z-direction, which represents flow in the opposite direction. Herein, turbulent kinetic energy is considered to be 1.473 m²/s², and the dissipation rate is 1337.05 m²/s³, showing the initial intensity of turbulence and its proper dissipation, respectively. These are the very critical parameters in setting the simulation up accurately for the k-epsilon model in order to effectively analyse the characteristics of aerodynamics and flow. Here, the setup of such parameters gives a good balance between computational efficiency and accuracy; therefore, indeed these are reliable to perform the simulation.

3.4 Computational Domain

This study involves the design of the Ahmed Body in square-back shape and one that features dimples on its sides. An integral part of this work is also the establishment of the computational domain boundaries, that is the finite region within which all flow simulations will be done. The geometry of the Ahmed Body and the dynamic of the flow around it defines the size and shape of the computational domain.

In the context of Computational Fluid Dynamics (CFD) studies, it is essential that the boundaries of the computational domain be placed far enough from the object being studied so that boundary effects are minimized, and accurate results are achieved. Following standard CFD practices (e.g., Versteeg & Malalasekera, 2007), the inlet boundary is placed many body lengths upstream while the outlet boundary is set far enough downstream to allow the flow to develop and stabilize. The lateral and upper boundaries are established at a sufficient distance to prevent any restriction of the flow surrounding the Ahmed Body.

This study is conducted mainly to determine the skin friction over the sides and to analyze the pressure influence on the rear part of the Ahmed Body. The computational framework is devised to closely replicate the external flow conditions by incorporating appropriate boundary conditions, including velocity inlets, pressure outlets, and no-slip walls. These settings allow for a realistic representation of the aerodynamic behavior, thus providing an accurate measure of the effects due to side dimples. Optimizing the dimensions of the computational domain and the placement of boundaries can achieve a balance between simulation accuracy and computational efficiency.
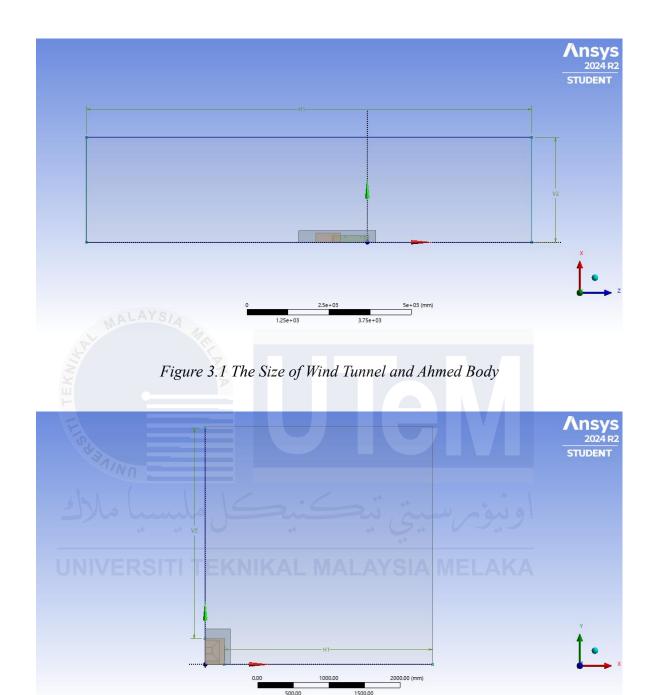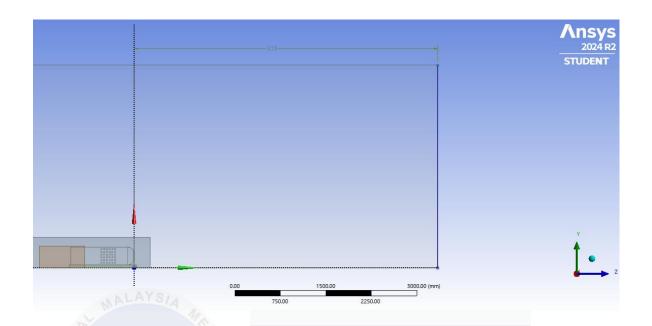
Figure 3.1 The Size of Wind Tunnel and Ahmed Body

Figure 3.2 Front View of Wind Tunnel

Figure 3.3 Distances from Front of Ahmed Body to Front Wind Tunnel

Table 3.1 Analysis Configuration

Velocity Speed (m/s)	**19.82**
Analysis Type	Steady State
Meshing near body size element	Element size: 0.165mm
Meshing boundary layer (Global)	First Layer Height: 0.015

The size and configuration of the wind tunnel are sufficient, relative to the Ahmed body in Figure 3.1, to minimize wall effects such that flow will be fairly uniform. Its cross-sectional area appears large enough that excessive blockage should be negated, but this should be substantiated through a calculation of the blockage ratio (Ahmed body frontal area divided by the wind tunnel cross-sectional area).

The meshing strategy followed with a near-body element size of 0.165 mm and first boundary layer height of 0.015 mm looks satisfactory for the resolution of boundary layer effects, although a grid independence study is required to confirm the same. A more

detailed explanation of the turbulence models adopted and validation with experimental data would lead to further establishing the credibility of the analysis.

3.4.1 Data Collection
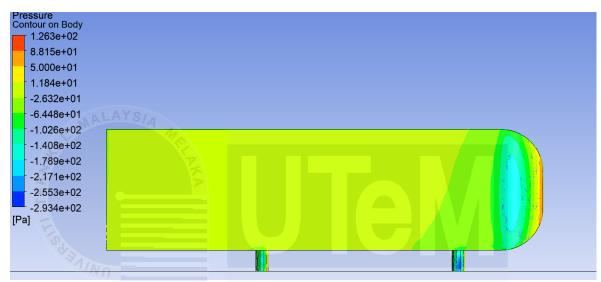
Figure 3.4 Pressure Coefficient Contour

The velocity contours and vector diagrams give graphical illustrations of the flow around the body. The flow separation and re-circulation can be recognized this way. Aerodynamics of the motion are quantified directly by the overall drag coefficients. The effect dimples create of diminishing the surface friction needs to be analyzed through local skin friction coefficients. Without this information, the significance of the effect of dimples on the aerodynamics of the Ahmed body cannot be understood.

3.4.2   Analysis Methods

Figure 3.5 Velocity Streamline

Analysis of data includes comparison analysis, flow visualization, and statistical analysis. The various dimple arrangements have their comparison patterns wi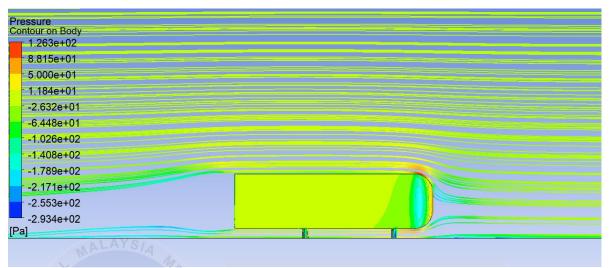th the standard Ahmed body for us to identify the patterns and identify improvements in aerodynamic performance in place. Streamlines and vortex identification are very helpful for a better prediction of flow mechanisms. The statistical approach determines importance regarding disparities observed and assures the robustness of findings. These tools of analysis will give an insight over the effect of side dimples on the aerodynamics of the Ahmed body in optimisation of designs.

3.5   Post Processing

After finishing the post-processing stage , the output for the Analysis Wind Tunnel earlier including the velocity profiles, Reynolds number, temperature distributions and pressure fields, will be used in this stage to analyze aerodynamic effects of side dimples on the modified models. The main goals for this analysis is to quantify the effect of the side

dimples on aerodynamic performance using CFD. For more specifically, this analysis aims to:

i. Quantify the Effect of Side Dimples on Skin Friction Drag

Skin friction drag is mainly influenced by the flow behaviour over the surface of the Ahmed body by using Reynolds number formula:

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

Where V is the velocity of the flow over the surface, L is the length of the dimple size or body length, ρ is the fluid density, and μ is the fluid's dynamic viscosity, the effect of side dimples on flow characteristics can be assesed to. Post- processing is focused on regions of the benchmark Ahmed body and also the modified Ahmed body which have the side dimples to determine the variation in skin friction coefficient.

ii. Velocity on Ahmed body

The calculation of the velocity with the use of the Reynolds number follow the attainment of the necessary parameters like fluid properties and dimensions, during the analysis in aerodynamics. The formula for the Reynolds number allows the determination of the value of the velocity, V, as:

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

Whereas, the Reynolds number (Re) is gven as 1.38×10^6, which is characterisitic of the aerodynamic flow conditions, while the dynamic viscosity (μ) is provided as $1.837 \times 10^{-5} \frac{kg}{m} . s$. The fluid density (ρ) is $1.225 kg/m^3$ is typical for air at standard conditions. And finally the characteristic length (L) is 1.044m, is corresponding to the ahmed body length.

$$V = \frac{(1.38 \times 10^6)(1.837 x 10^{-5})}{(1.225)(1.044)}$$

The velocity of the air is calculated to be approximately 19.82m/s.

3.6   Result Comparisons

The results of the simulation will be compared between the standard Ahmed Body and the modified Ahmed body that had side dimples. This study will focus on evaluating the skin friction that act on the Ahmed body as well as examining the drag coefficient (Cd), lift coefficient (Cl) and moment coefficient (Cm). These coefficients will be utilized to construct graphs, which will be analyzed and discussed in detail in the Chapter 4.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Preliminary

This chapter will present the simulations and results obtained in the study using Ansys CFD (Fluent flow) and a comparison between two configurations which is the modified model and benchmark model to determine the difference of improvement in drag reduction. The main goal is to assess the aerodynamic characteristics in the redesigned model as distinct from the basic design. Performance parameters such as drag coefficient, skin friction and flow characteristics of the Ahmed body are thus the main concern of the analysis. The general aerodynamic efficiency of the vehicle and aerodynamics that forms the Ahmed body design are discussed considering the findings.

4.2   Ahmed body model

CATIA is crucial in the designing process of Ahmed body, which offers a comprehensive, friendly environment for modelling and refinement. Advanced CAD tools within the application provide an engineer with the ability to promptly define an initial sketch, establish a base model, and make precision adjustments for performance improvement.

The fully developed parametric modelling capability of CATIA allows for quick changes with maintained consistency and accuracy in the design. It allows for the easy optimization of the model toward high standards of performance in aerodynamics.

The intuitive interface and detailed visualization through CATIA make it easier to analyse and validate the design before running the simulations. From the use of CATIA, it would be possible to ensure that the model of Ahmed body is accurate and ready for further test and analysis in this research.

4.2.1 Benchmark drawing of Ahmed Body

Results from these benchmarks are significant in this study as they form the trusted basis against which performance comparison and validation shall be done on the modified model. The Ahmed square-back model was thus used in this experiment as the reference standard. To make it more like a real front profile, an edge fillet radius of 100 mm was given along its front edges to the Ahmed body. In so doing, not only has the aerodynamic realism of the model been increased but a common framework is also established in comparing further surface modifications - say, the addition of dimples - to realize an improvement in the aerodynamic performance.
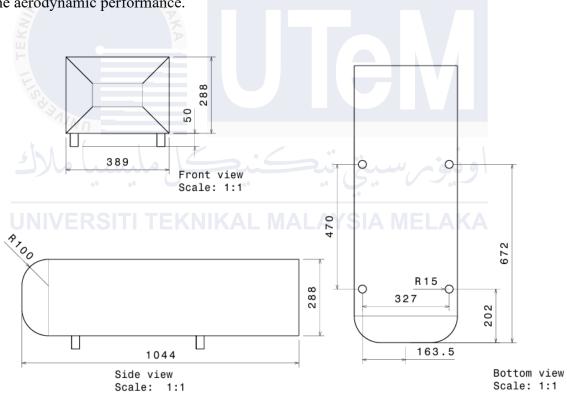

Figure 4.0 Benchmark Model

This investigation uses the Ahmed Body model, described by (Minguez et al., 2008), as a starting point and with further modification to fit the purpose of this investigation. Both the benchmark and modified models do not feature the typical 20-degree slant angle at the

71

rear but use a square-back configuration instead. The leading edges at the front are also rounded with a 100 mm fillet radius to better model the real vehicle geometries.

All models used during this investigation, including those featuring different dimple configurations, originate from this modified Ahmed Body design. The Benchmark model, however, excludes all but two of these modifications—the removal of the slant angle and the adjustment of its front edge. The drag coefficient obtained using this Benchmark model is then used as the baseline against which the other modified designs are compared regarding their aerodynamic performance.

4.2.2   Dimples on Side of Ahmed Body (Modified)

To minimize the flow separation caused by the boundary layer, the present study considers the effect of dimples on the side surface of the Ahmed Body. Dimples induce local turbulence, which delays flow separation and, as a result, diminishes the area of the wake region, hence reducing the pressure drag. The outcome is an improvement regarding the benchmark Ahmed Body without dimples, as seen by enhanced pressure retention along the sides and therefore smooth velocity streamlines, reducing the coefficient of drag. This agrees with previous studies, which utilized dimpled surfaces in enhancing attachment and drag reduction for bluff body flows (Latif et al., 2019)
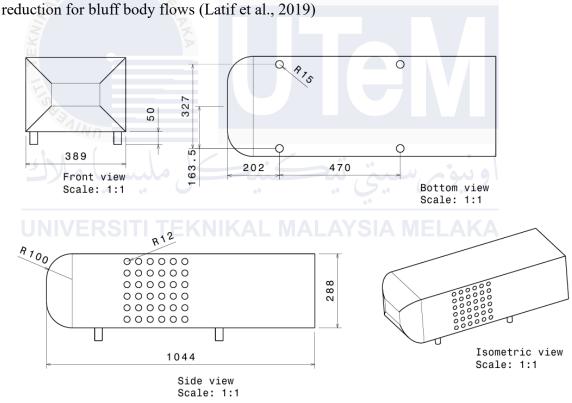
Figure 4.1 Modified Model
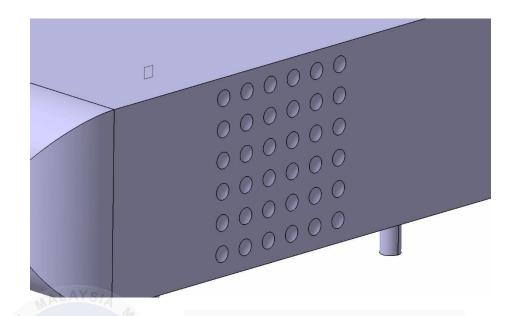
Figure 4.2 Side Dimples on Modified Model

The Ahmed Body side dimples were designed with a diameter of 40 mm and the offset distance from the plane to revolve the dimple set at 16 mm. These are the dimples trying to reduce the boundary layer separation along the side surfaces of the Ahmed Body. Its placing distance has been calculated as can be seen from the distance of the front end of Ahmed Body, at 301.2 mm, and the distance to the rear end at 505.8 mm, as shown in Figures 4.1 and 4.2.

4.2.3 Meshing

A set of preliminary simulations were carried out in order to establish the best mesh size of the Ahmed Body with a dimpled plate, prior to the actual launching of the Ansys CFD Flow Fluent, following the description made in this section a total number of six simulations have been computed to attain an element count ranging from 3 million to 18 million. These were based on a different size for the elements of major components that comprised the model, namely, the body itself, and the wind tunnel dimensions-symmetry, input, output, walls, and road-to get a compatible and accurate grid sensitivity analysis.

Meshing tools from Ansys Fluent, such as 2 Face Sizing and 3 Body Sizing, have been used here. However, detailed mesh generation was focused on the step sizing of Face Sizing 2. No adaptive sizing is used, and the growth rate is by default taken as 1.2. This number defines in which ratio the size of adjacent elements should grow. For all computations, the maximum size specified is 14.131 m. Further, defeaturing was turned on and off to wipe out any tiny geometrical details. The "Capture Curvature" and "Capture Proximity" options were active. These settings make sure proper resolution of geometric curves-meshing captures proper curvatures-while also capturing the details in such locations wherein the surfaces are relatively not far from each other-something especially important for adequate representation of complex aerodynamics.

The program will control the automatic setting for inflation. Inflation options are used to capture the boundary layer around the Ahmed Body by using the first aspect ratio of 5. This is, therefore, good for capturing the boundary layer since inflation layers refine the mesh around the wall to effectively capture the flow gradients. The only element size changed within the simulations was Face Sizing 2 (Ahmed Body) to increase the resolution of the mesh and consequently the accuracy of the results. Hence, this will, therefore, enable

a systematic study to be done on the quality of the mesh and the suitability of this kind of

aeromechanical analysis.

Table 4.0 Benchmark Meshing Sizes

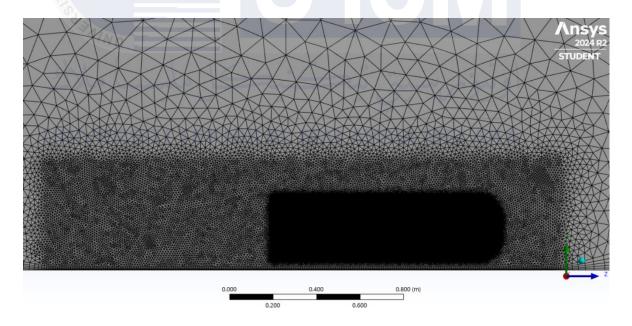| Benchmark	Element size (Body)	Element Count
3,000,000	0.0038	3,029,694
6,000,000	0.00232	6,018,248
9,000,000	0.0017	9,094,449
12,000,000	0.00144	12,093,150
15,000,000	0.00128	15,044,826
18,000,000	0.001165	18,075,336

Figure 4.3 The mesh of Benchmark Ahmed Body (3million-18million)

4.3   Ansys CFD Fluent Flow

It is important to emphasize in the present report the significant role played by Ansys Fluent, a multidimensional computational fluid dynamics (CFD) software. Ansys Fluent has been central to our research and has enabled modelling in fluid dynamics. This software allows much more than just fundamental simulations; it enables a detailed analysis of various fluid flow phenomena. One of the most notable features is the advanced yet user-friendly interface of Fluent.

Which greatly guides us through the entire CFD process, from setting up simulations to analysing results. This optimized workflow, brought together in a single interface, significantly enhances efficiency and accessibility, making Ansys Fluent a very valuable tool in our research. Using Ansys Fluent for the simulation of our study provides a powerful mechanism to obtain and analyse results to increase the overall success of our research.

Table 4.1 Benchmark Setup

Boundary conditions	Benchmark
Velocity inlet, m/s	19.82
Pressure outlet, Pa	0
Reynolds number	1.38×10^6
Fluid density, ρ	1.225

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

$$V = \frac{(1.38 \times 10^6)(1.837 x 10^{-5})}{(1.225)(1.044)}$$

$$V = 19.82 m/s$$

Figure 4.4 Velocity Calculation

In this analysis, boundary conditions were carefully adjusted based on the calculated values shown in Figure 4.5 to ensure accurate simulation of flow dynamics around the Ahmed Body. Simulation results were compiled in Excel, organized into tables and graphs, and refined to emphasize key parameters such as drag coefficients and flow separation metrics. Comparisons were made across simulations with mesh densities ranging from 3 million to 18 million elements, allowing the identification of optimal conditions for the side dimples, as depicted in Figure 4.6. This systematic process ensured reliable results and deeper insight into the aerodynamic effects of the modifications.
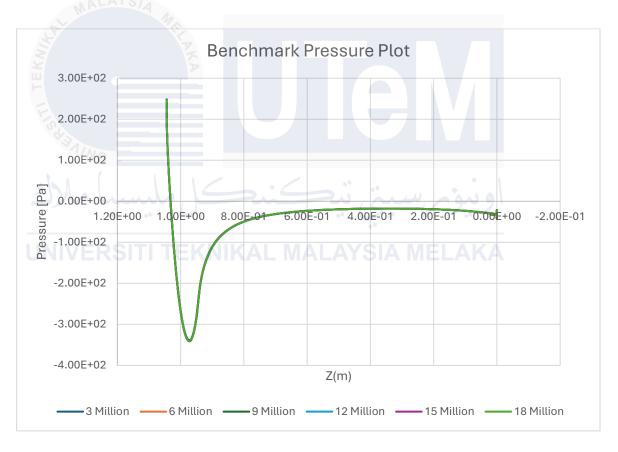
Figure 4.5 Pressure Plot 3 Million to 18 Million Meshing

4.3.1 Mesh and Ansys for 12 million Modified Ahmed Body

Comparing the different pressure plot graphs between 3 million to 18 million elements, the result showed the optimum mesh element size at 12 million elements. Therefore, the mesh element size of 0.00158 mm was used over the modified Ahmed Body's side dimples for capturing appropriate results in that region. Other regions within the geometry had meshing conditions that remained the same since most of the major focus was set upon the Ahmed Body itself to provide an improvement in the precision of results within the given aerodynamics. This eventually gave the setting desired that led to approximately 12.02 million elements in total for the Modified Ahmed Body.
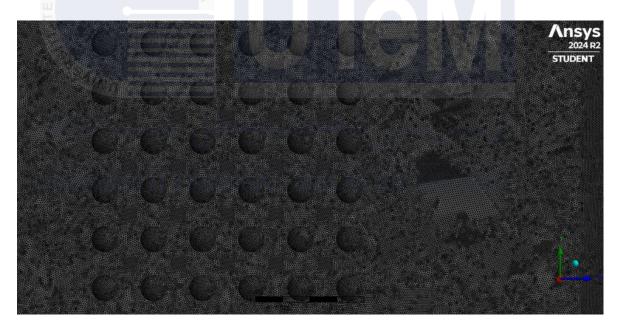
Figure 4.6 Mesh view for dimple on Modified Ahmed Body

Then, the solution phase had to be undertaken in ANSYS Fluent to run the simulation. The simulation was done with the same settings taken by the benchmark of 12 million elements with a velocity of, V= 19.82m/s which corresponds to a Reynolds number of (1.38×10^6) as considered for both the benchmark and the modified model. This was important in ensuring valid data on the reduction of drag during post-processing. The same procedure was done for the benchmark model of 12-million-element in order to directly compare the degree of drag coefficient reduction between the benchmark and modified models and also to make the skin friction analysis.
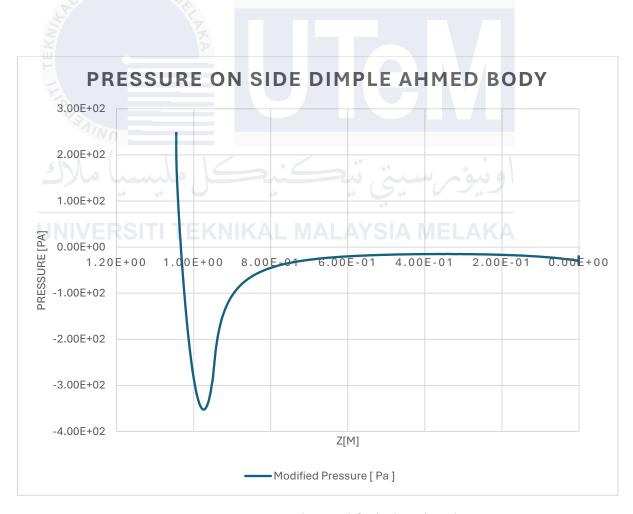
Figure 4.7 Pressure Plot Modified Ahmed Body

4.4 Benchmark vs. Modified Ahmed Body

4.4.1 Pressure Plot on Side Surface

During analysis, Benchmark and Modified results were compared by integrating their pressure data and further visualizing them against their respective Excel data tables shown in Figure 4.7. From this combined data, the observation comes out that the pressure characteristic differences of the two are negligible, with deviations running below 1 Pa through most of the range.

The above observation flags the fact that pressure distribution over Ahmed's body is relatively the same whether he is at rest or in any other condition within a wind tunnel. This would mean the high level of agreement measured from the two graphs essentially underlines the reproducibility and dependability of both experimental conditions and equipment. The consistency in the results for pressure tends to ensure the accuracy of the applied methodology of analysis, which can reproduce stable and dependable results even in changing conditions.
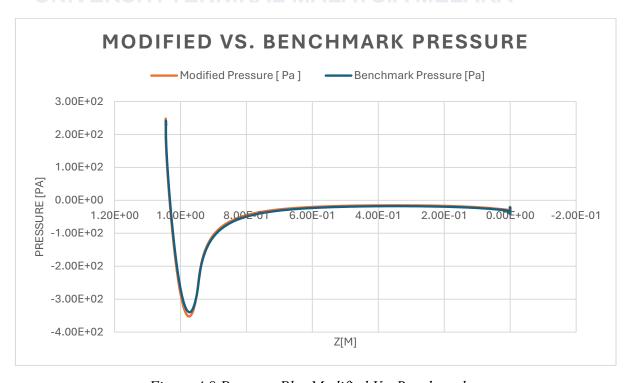
Figure 4.8 Pressure Plot Modified Vs. Benchmark

4.4.2 Coefficient Drag (Cd)

In this analysis, the drag coefficient data from both the Benchmark and Modified bodies were put together, along with their respective Excel data sheets, for a side-by-side comparison, as shown in Figure 4.8. A glance at the combined information reveals a marked difference in air resistance, or drag coefficient, which each body faced. That is, the Benchmark model had a drag coefficient, (Cd) of 0.306, while the Modified model had a drag coefficient, (Cd) of 0.277. This constitutes a reduction in the drag coefficient by approximately 8.46% over the Benchmark. The result of the analysis implies the success of the project where adding the side dimple yielded a measurable drag reduction on the Ahmed body. Improvement in drag relates to the improvement in the flow conditions, which sustains the premise of this research-that the side dimple was effective in bettering the airflow of the body within the wind tunnel.
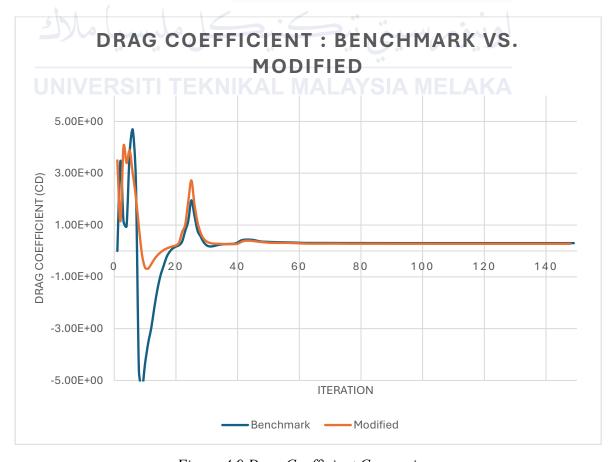
Figure 4.9 Drag Coefficient Comparison

4.4.3   Coefficient of Lift (Cl)

The graph represents the comparative (Cl) for 160 iterations in two configurations. The x-axis shows the iteration count, while the y-axis is given by Cl over a range from -60 to 20. In the Benchmark case, the blue line reflects stability with minor fluctuations, which indicates consistent aerodynamic performance. The Modified case is an orange line that shows large initial instabilities with high variations in the value of Cl. After approximately 40 iterations, the Modified case stabilizes around zero to find its steady-state performance. This analysis gives a view on what modifications can do to transient behaviour, while iterative optimization is necessary to get as good stability as in the Benchmark.
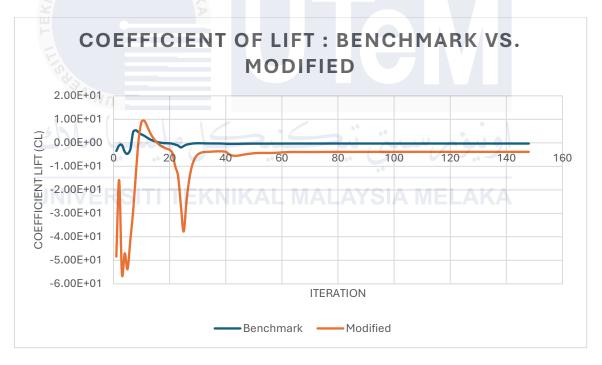
Figure 4.10 Coefficient of Lift Comparison

4.4.4   Coefficient Moment (Cm)

The graph on Coefficient of Moment presents the moment coefficient dependence on 160 iterations for two configurations. In the former, on the x-axis it takes iteration count and on the y-axis, it takes the values of (Cm) between -0.45 to 0.05. There are variations at first before leveling off to a negative 0.2, despite the fact that Benchmark has a fluctuating performance after twenty runs of experiencing some range variability. The Modified case shows greater volatility of the initial runs but, at the same time, it stabilizes as close as possible to -0.2 of the times axis. This further analysis implies that even though the introduced modification has inherent transient instability the overall performance is as good as the Benchmark at a more stable state, therefore the modification can be considered to be aerodynamically reliable once optimized.
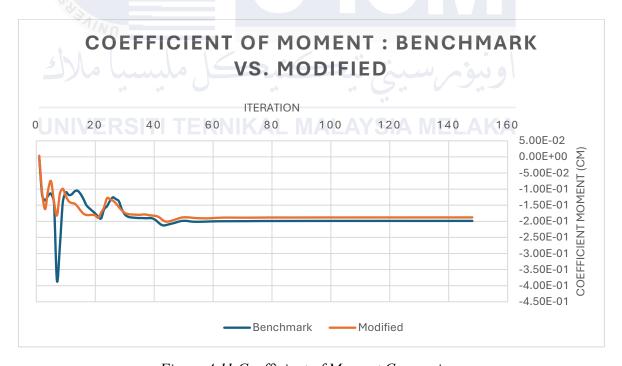
Figure 4.11 Coefficient of Moment Comparison

4.4.5 Skin Friction

The graph shows a comparison of the skin friction coefficient (Cf) on the benchmark Ahmed body and its modified version with a side dimple surface. Both profiles follow similar trends characterized by high peak values close to the leading edge (Z=0), followed by the overall skin friction. The reference configuration shows a marginally higher peak value at about 0.014, whereas for the modified body, it is 0.0135, which suggests a marginal reduction in localized drag due to the dimpled design. Further downstream, the profiles converge, indicating aerodynamic stability in both configurations. The dimpled surface performs uniformly, meaning it can realize small improvements in aerodynamics without largely affecting the flow characteristics.
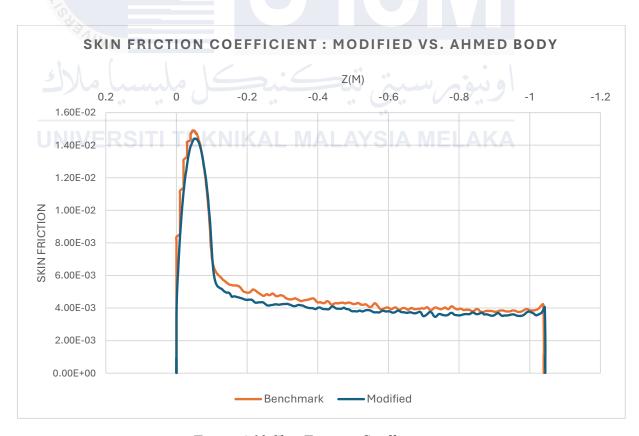
Figure 4.12 Skin Friction Coefficient

4.5 Statistical Analysis of Pressure Distribution

The pressure distributions of the Benchmark and Modified models have been compared to evaluate the improvement in performance and validate design changes made in the models. Statistical tests were carried out in order to test whether modification made resulted in significant improvement. A comparison of means using both a parametric test (t-test) and a non-parametric approach (Wilcoxon signed-rank test) was carried out, and statistical summaries were given to underline the most important differences between the models.

The pressure values for both the models have been extracted along the -axis. The t-test compared the means of the pressures, and the t-statistic was computed as:

where and are sample means, and are sample variances, and, are sample sizes. In case degrees of freedom were computed using Welch's formula and the corresponding p-value defined statistical significance of the difference. The test of Wilcoxon signed-rank, that rank the absolute differences between each couple of paired observations, was used as a robust alternative if the assumptions concerning normality had not been fulfilled.

Summary statistics: mean pressures, medians, standard deviations and pressure ranges are given in the following Table 4.2.
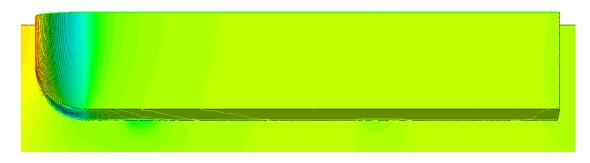
Table 4.2 Statistical Analysis

Statistic	Modified Model	Benchmark Model	Value
Median Pressure (Pa)	-5.61E+05	-7.81E+05	-
Mean Pressure (Pa)	-2.66E+05	-6.49E+05	-
Standard Deviation (Pa)	2.40E+06	2.40E+06	-
T-test p-value	-	-	0.4090
Wilcoxon p-value	-	-	0.1953

The analysis of the results revealed that the mean and median pressures desirability for the modified and benchmark system are different but not significantly different. Mean of the modified pressure is -266000 Pascals in both cases while medians are -561000 and -781000 Pascals for the benchmark and the original pressure respectively. Both systems are notably variable with standard deviations of about 2.4M Pascals, implying that the pressure readings are widely distributed.

In addition, the t-test (p = 0.4090) and the Wilcoxon test both show that the systems are statistically indistinguishable, (p = 0.1953). These implications suggest that the modified system is nearly equivalent to the benchmark thus requiring minimal further fine tuning. Thus, if the precision is important, possible subtle variations must be explored as the variance is very broad. All in all, the findings make it possible to claim that the modified system performance is similar to that of the benchmark, which guarantees its efficient work.

The pressure distributions in case of Benchmark and Modified models are represented by Figures 4.13.
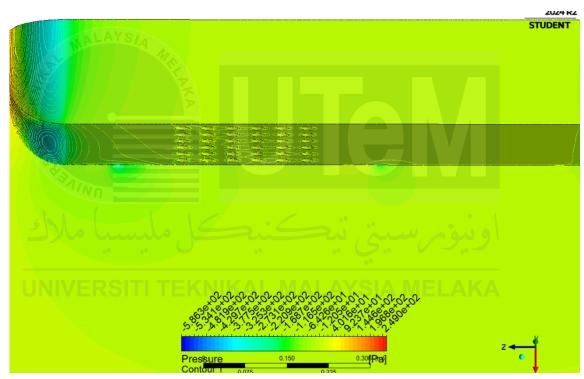
Figure 4.13 Benchmark Vs. Modified Pressure Distribution

The following chart plots against each other the benchmark pressure and the modified pressure of the eight planes, with error bars showing the standard deviations and trend lines that provide overall trends. The error bars, indicative of variation, remain fairly constant throughout for both modified and benchmark pressures and, therefore, demonstrate largely similar levels of variabilities between the planes.

The trend lines show slight negative slopes for both pressures; modified pressures are less negative compared to the benchmark. The modified pressure in Plane P1 is higher than the benchmark and shows an increase as brought about by the modifications. From P2 through P8, the modifications seem to decrease the negative pressure. This chart thus shows that, overall, modifications to the aircraft have resulted in a net positive alteration to the pressure conditions, with variability remaining constant among the planes.
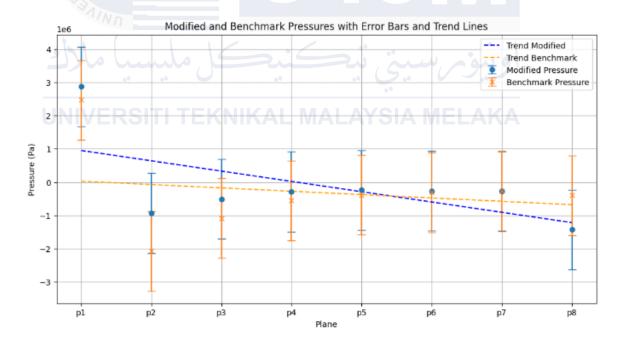
Figure 4.14 Error Bars and Trend Lines

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1   Preliminary

This chapter summarizes the major conclusions derived from the research and gives a view on possible future directions the research might take. The work carried out on the Ahmed body, regarding the effect of side dimples, has provided the necessary understanding in the context of aerodynamic optimization for vehicles. In this regard, CFD simulations have shown succinctly that surface modifications like the addition of side dimples enhance performance in vehicles and reduce aerodynamic drag. The obtained results underline the importance of innovative design approaches in ensuring improvements concerning fuel efficiency and sustainability within the sector.

5.2   Conclusion

This study investigating the impact of side dimples on the aerodynamics efficiency of Ahmed body in terms of skin friction reduction, total drag coefficient, and pressure distribution using Computational Fluid Dynamics (CFD).

The results demonstrate the considerable enhancement of the aerodynamic performance and the dimple-treated Ahmed body. The drag coefficient decreased by about 9.48%, which implies improved airflow patterns and less air resistance. In addition, the skin friction coefficient also exhibited a slight decrease in the vicinity of the front edge, thereby

validating the role of the dimpled pattern in mitigating localized drag and without further compromising the aerodynamic stability in the downstream direction.

In pressure distribution across the field, although average and median pressures of the modified and benchmark systems were found to be dissimilar, statistical tests failed to establish any considerable difference, hence the performance of the modified design is essentially the same as that of the benchmark. The wide variation in pressure readings indicates space for further improvement should accuracy be an absolute requirement.

In conclusion, the present study validates that adding side dimples on the Ahmed body is a promising method for improving aerodynamic performance in the form of drag reduction. The results are valuable inputs for framing future research activities in aerodynamic design and optimization.

5.3   Impact Research

Based on the current work of research, huge implications for further studies, as well as those for the automotive industry, can be made. The obtained results have shown a large potential of side dimples on the reduction of aerodynamic drag, directly related to fuel efficiency and performance improvement. Considering a more general economic aspect, this new automotive design feature will bring value not only by decreasing the production costs but also for consumers, contributing to helping the Earth accomplish its environmental objectives-for example, decreasing greenhouse gas emissions.

Side dimples open a completely new direction for vehicle aerodynamics optimization, able to develop an innovative design optimization methodology based on the principles of sustainable development. In other words, further paths for research into improvements in the aerodynamics of the vehicle are open, giving grounds for further

optimizations toward better performance and higher sustainability within the automotive industry.

5.4   Limitations

Although this study has gained insights of aerodynamic characteristics of side dimples of the Ahmed body, there are few limitations inherent in this study that must be recognized. Firstly, the study was carried out using basic geometries and models for which the CFD models were developed, namely the Ahmed body which while useful for providing validation data points may not be an exact representation of full realistic car geometries. It was also noted that using simulation models of rate geometries and incorporating more appendages to sides of vehicles would give a more detailed perception of the utility of side dimples in actual vehicles since the present models provide average results.

Secondly or rather limitation the CFD analysis depended on several assumptions such as boundary condition, turbulence models and meshing resolutions. That is why, even though significant efforts were devoted to the validation of CFD results, such simulations may be a function of these parameters within a certain small range. Further work should be aimed at exploring the experimental verification by employing wind tunnel data or full field evaluation to support credibility of the conclusions in dynamic conditions for building and structure design.

One of the limitations of this work is that it only investigates one modification, the side dimple. Moreover, the present work demonstrated the enhancement in the aerodynamic performance, but other parameters including shapes and textures of vehicle body, or interaction of various modifications might help in the improvement of aerodynamic efficiency. Based on the present work, more research could be done to analyze a combination

of the side dimples with other aerodynamics, including vortex generators or active flow controls.

Furthermore, the computational time needed when simulations are run at high mesh density may not allow researchers to employ those techniques on large vehicle system designs. In this study, high quality meshes combined with a variety of simulations have been used, yet the computational demands of full-size vehicles result in the practical problem of accessibility of the approach For real-time applications, the scope may be fairly restricted.

Last but not the least, the long term sustainability and economic factors regarding sideward protrusions also known as automobile side dimples are inconclusive. While the goal of this study is simply to evaluate the short-term visa aerodynamic improvements, the application of such modifications would require the investigation of the long-term consequences of the changes in question on car maintenance, the sturdiness of materials used and the cost of the lifecycle of a vehicle before they are produced en masse.

5.5   Recommendations of Future Work

In this study numerous advances have been made to learn more about the side dimples in relation to aerodynamics; however, there is still many areas that need further research in order to realize full potential of such changes. There exist a challenging direction to develop further: with measurement parameters other than and combined with the size, the depth and shape of the dimple and arrangement. Further work may be needed to investigate the details of these parameters to better ascertain the preferred array geometry for drag reduction. These findings would offer a greater insight of how the changing in dimple geometry influences the aerodynamic capability.

Besides, the results obtained with CFD simulation must be checked with experiments, especially with wind tunnel tests and field measurements. These tests are

critical for verifying the CFD result and evaluating the practicality of the side dimples concept in the actual automobiles.

Another area that can be developed in further works is an extension of the study to the full-scale vehicle models. Scale models would generate more accurate results on the impact of side dimples on aerodynamics in general drive since full-size models come with other characteristics arising from the size and interrelations of actual vehicle functioning.

Furthermore, it is possible to investigate the benefits of combined application of side dimples and other forms of external excitation, including, for instance, vortex generators or active flow control systems. Possibly, the synergy of these technologies could yield a sum total effect of improving the aerodynamics of a car.

Last but not the least; the side dimple Embossing on the vehicle should be discussed about the long term service life of these features and the economic feasibility of the entire process. Long-term studies should identify further changes' benefits and detriments that have occurred to/on the car life cycle, maintenance, and cost-efficiency. Such studies would be significant when assessing the viability of side dimple application on large scale industries in automotive sector to identify their sustainability economically in future.

So, it can be concluded that the given study demonstrated a successful attempt to depart the new area of side dimples and similar surface shapes While the given study revealed the concept of the side dimples applied to the certain car models and demonstrated promising outcomes, these research directions can be expanded to further discover the effectiveness of side dimples and similar surface shapes for optimizing the car aerodynamics. This will assist in decision making on the practical implementation of the advanced technologies in the automotive industry to enhance efficient and environmentally sustainable transport technology designs.

94

REFERENCES

Aabid, A., & Khan, S. A. (2021). Investigation of High-Speed Flow Control from CD Nozzle Using Design of Experiments and CFD Methods. *Arabian Journal for Science and Engineering*, *46*(3), 2201 – 2230. https://doi.org/10.1007/s13369-020-05042-z

Abdalkarem, A. A. M., Fazlizan, A., Muzammil, W. K., Lim, C. H., Ibrahim, A., Wong, K. H., & Kazem, H. A. (2024). The Effect of Various Wedge Flap Configurations on the Performance of Wind Turbine Airfoil. *Iranian Journal of Science and Technology - Transactions of Mechanical Engineering*. https://doi.org/10.1007/s40997-023-00743-w

Akhter, Md. Z., Jawahar, H. K., Omar, F. K., & Elnajjar, E. (2024). Performance characterization of a slotted wind turbine airfoil featuring passive blowing. *Energy Reports*, *11*, 720 – 735. https://doi.org/10.1016/j.egyr.2023.12.027

AL-bonsrulah, H. A. Z., Danook, S. H., Alshukri, M. J., Ahmed, A. M., Raja, V., Veeman, D., & Al-Bahrani, M. (2023). CFD modeling of a horizontal wind turbine by utilizing solar nozzle for power production. *International Journal of Low-Carbon Technologies*, *18*, 31–37. https://doi.org/10.1093/ijlct/ctac127

Al-Kayiem, H. H., Lim, D. C., & Kurnia, J. C. (2020). Large eddy simulation of near-wall turbulent flow over streamlined riblet-structured surface for drag reduction in a rectangular channel. *Thermal Science*, *24*, 2793 – 2808. https://doi.org/10.2298/TSCI190909059A

Al-Kayiem, H. H., Othman, A. R., Abed, A. B., & Tukkee, A. M. (2023). Experimental Evaluation of Drag Reduction in Pipe Flow Using Streamlined Turbulence Modifiers. *Lecture Notes in Mechanical Engineering*, 237 – 248. https://doi.org/10.1007/978-981-19-1939-8_21

Allarton, R., Yao, J., Clifford, T., Hitchborn, B., Parker, L. J., & Shaw, J. (2020). Surface flow modification of aerofoils for automotive racing car applications. *International Journal of Modern Physics B*, *34*(14n16), 2040096. https://doi.org/10.1142/S0217979220400962

Azahar, M. A. R., Ishak, I. A., Sulaiman, S., Jaat, N., Zulkifli, A. F. H., Sukiman, S. L., Osman, K., & Darlis, N. (2023). The Effect of Different Reynolds Number On Solid Sphere Using CFD and Its Verification. *International Journal of Integrated Engineering*, *15*(1), 96 – 105. https://doi.org/10.30880/ijie.2023.15.01.009

Azlan, F., Tan, M. K., Tan, B. T., & Ismadi, M.-Z. (2023). Passive flow-field control using dimples for performance enhancement of horizontal axis wind turbine. *Energy*, *271*, 127090. https://doi.org/10.1016/j.energy.2023.127090

Ballerstein, N., & Horst, P. (2023). Manipulation of the aerodynamic behavior of the DrivAer model using dimple patterns. *JOURNAL OF WIND ENGINEERING AND INDUSTRIAL AERODYNAMICS*, *233*. https://doi.org/10.1016/j.jweia.2022.105216

Chen, C., Lu, Y., & Xue, C. (2021). Drag reduction mechanism of vapour-assisted liquid membrane lubrication on texture surface with elevated temperature. *Lubrication Science*, *33*(6), 358–368. https://doi.org/10.1002/ls.1558

Elliott, J., Smith, L., Lyu, B., & Smith, B. L. (2024). The combined influence of spin and roughness frequency on sphere aerodynamics. *Experiments in Fluids*, *65*(4), 60. https://doi.org/10.1007/s00348-024-03794-0

Ghani, J. A., Mohd Nasir, F. F., Rahman, H. A., Hakim Wan Zamri, W. F., Kasim, M. S., & Muhamad, S. S. (2021). Computational fluid dynamic analysis on tribological performance under hydrodynamic lubrication of dimple textured surface produced using turning process. *Wear*, *477*, 203898. https://doi.org/10.1016/j.wear.2021.203898

Hossain, M. Z., & Floryan, J. M. (2020). On the role of surface grooves in the reduction of pressure losses in heated channels. *Physics of Fluids*, *32*(8). https://doi.org/10.1063/5.0018416

Ismail, M. P., Ishak, I. A., Samiran, N. A., Mohammad, A. F., Mohd Salleh, Z., & Darlis, N. (2022). CFD Analysis on the Effect of Vortex Generator on Sedan Car using ANSYS Software. *International Journal of Integrated Engineering*, *14*(1), 73 – 83. https://doi.org/10.30880/ijie.2022.14.01.008

Kashyap, V., Mittal, P., Arora, B. B., Arora, A., & Bhattacharjee, S. (2021). Computationally Analyzing the Impact of Spherical Depressions on the Sides of Hatchback Cars. *SAE INTERNATIONAL JOURNAL OF COMMERCIAL VEHICLES*, *14*(1), 111–123. https://doi.org/10.4271/02-14-01-0008

Krishnan, A., Al-Obaidi, A. Sh. M., & Hao, L. C. (2023). A comprehensive review of innovative wind turbine airfoil and blade designs: Toward enhanced efficiency and sustainability. *Sustainable Energy Technologies and Assessments*, *60*. https://doi.org/10.1016/j.seta.2023.103511

Ladommatos, N. (2021). Influence of air rifle pellet geometry on aerodynamic drag. *Proceedings of the Institution of Mechanical Engineers, Part P: Journal of Sports Engineering and Technology*, *235*(4), 257 – 276. https://doi.org/10.1177/1754337119831107

Lateb, M., Dghim, M., & Fellouah, H. (2022). Numerical Investigation on the Effect of Lateral Skirts Device on the Flow Dynamics around a Semi-trailer Truck. *SAE International Journal of Commercial Vehicles*, *16*(1). https://doi.org/10.4271/02-16-01-0001

Lewthwaite, M. T., & Amaechi, C. V. (2022). Numerical Investigation of Winglet Aerodynamics and Dimple Effect of NACA 0017 Airfoil for a Freight Aircraft. *INVENTIONS*, *7*(1). https://doi.org/10.3390/inventions7010031

Liu, B., Xu, X., & Pan, D. (2022). Drag reduction configuration analysis of the grooves on amphibious vehicles walking mechanism. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, *236*(11), 5994–6006. https://doi.org/10.1177/09544062211065380

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*, 1–19. https://doi.org/10.1155/2022/3269604

Mao, X., & Wang, B. (2021). Spanwise localized control for drag reduction in flow passing a cylinder. *Journal of Fluid Mechanics*, *915*. https://doi.org/10.1017/jfm.2021.154

Mohd Basri, M. S., Kamarudin, K. M., Maidin, N. A., & Ab Rahman, M. H. (2020). Aerodynamic Drag Study of Time-Trial Cycling Helmets Using CFD Analysis. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, *72*(1), 21–31. https://doi.org/10.37934/arfmts.72.1.2131

Ng, J. H., Jaiman, R. K., Lim, T. T., Tay, C. M., & Khoo, B. C. (2020). Geometric Effects of Shallow Dimples in Turbulent Channel Flows at Re$\tau \approx$ 180 : A Vorticity Transport Perspective. *Flow, Turbulence and Combustion*, *105*(1), 83 – 122. https://doi.org/10.1007/s10494-020-00112-6

Olivucci, P., Wise, D. J., & Ricco, P. (2021). Reduction of turbulent skin-friction drag by passively rotating discs. *Journal of Fluid Mechanics*, *923*. https://doi.org/10.1017/jfm.2021.533

Panda, J. P., Handique, J., & Warrior V, H. (2023). Mechanics of drag reduction of an axisymmetric body of revolution with shallow dimples. *PROCEEDINGS OF THE INSTITUTION OF MECHANICAL ENGINEERS PART M-JOURNAL OF ENGINEERING FOR THE MARITIME ENVIRONMENT*, *237*(1), 227–237. https://doi.org/10.1177/14750902221100114

Qi, J., Qi, Y., Chen, Q., & Yan, F. (2022). A Study of Drag Reduction on Cylinders with Different V-Groove Depths on the Surface. *WATER*, *14*(1). https://doi.org/10.3390/w14010036

Qiu, H., Chauhan, K., & Lei, C. (2020). A numerical study of drag reduction performance of simplified shell surface microstructures. *Ocean Engineering*, *217*, 107916. https://doi.org/10.1016/j.oceaneng.2020.107916

Rafiqi Daniel Ramadhan Mariaprakasam, Shabudin Mat, Pakharuddin Mohd Samin, Norazila Othman, Mastura Ab Wahid, & Mazuriah Said. (2023). Review on Flow Controls for Vehicles Aerodynamic Drag Reduction. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, *101*(1), 11–36. https://doi.org/10.37934/arfmts.101.1.1136

Salinas, M. F., Botez, R. M., & Gauthier, G. (2021). New Numerical and Measurements Flow Analyses Near Radars. *Applied Mechanics*, *2*(2), 303 – 330. https://doi.org/10.3390/applmech2020019

Samiolo, M., & Verdin, P. G. (2022). Numerical modelling of a finless heat exchanger layout for electric vehicle application. *Applied Thermal Engineering*, *211*, 118506. https://doi.org/10.1016/j.applthermaleng.2022.118506

Siddiqui, N. A., & Agelin-Chaab, M. (2022). Detached Eddy Simulation of the 28° Ahmed Body at a Low Reynolds Number. *Volume 1: Fluid Applications and Systems*

97

(FASTC); Fluid Measurement and Instrumentation (FMITC); Fluid Mechanics (FMTC), *1*. https://doi.org/10.1115/FEDSM2022-87647

Siddiqui, N. A., & Agelin-Chaab, M. (2023). Effect of aspect ratio on the recirculation region of 35° Ahmed body. *Australian Journal of Mechanical Engineering*, *21*(2), 368 – 383. https://doi.org/10.1080/14484846.2020.1842307

Soyler, M., Ozalp, C., Polat, C., Saydam, D. B., & Yavuz, M. M. (2022). Experimental and numerical investigation of flow structure of grooved cylinder. *Ocean Engineering*, *264*, 112478. https://doi.org/10.1016/j.oceaneng.2022.112478

Svorcan, J., Wang, J. M., & Griffin, K. P. (2022). Current state and future trends in boundary layer control on lifting surfaces. *Advances in Mechanical Engineering*, *14*(7). https://doi.org/10.1177/16878132221112161

Tomlinson, S. D., & Papageorgiou, D. T. (2022). Linear instability of lid- and pressure-driven flows in channels textured with longitudinal superhydrophobic grooves. *Journal of Fluid Mechanics*, *932*, A12. https://doi.org/10.1017/jfm.2021.990

Uddin, G. M., Niazi, S. G., Arafat, S. M., Kamran, M. S., Farooq, M., Hayat, N., Malik, S. A., Zeid, A., Kamarthi, S., Saqib, S., & Chaudhry, I. A. (2020). Neural networks assisted computational aero-acoustic analysis of an isolated tire. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, *234*(10–11), 2561–2577. https://doi.org/10.1177/0954407020915104

Wang, C., DeGroot, C. T., & Floryan, J. M. (2024). Numerical simulation of drag reduction for turbulent flow in cylindrical annuli with axial corrugations. *Transactions of the Canadian Society for Mechanical Engineering*, *48*(1), 164–172. https://doi.org/10.1139/tcsme-2023-0034

Wang, J., Ma, W., Li, X., Bu, W., & Liu, C. (2019). Slip theory-based numerical investigation of the fluid transport behavior on a surface with a biomimetic microstructure. *ENGINEERING APPLICATIONS OF COMPUTATIONAL FLUID MECHANICS*, *13*(1), 609–622. https://doi.org/10.1080/19942060.2019.1630676

Wang, R., Cheng, S., & Ting, D. S.-K. (2020). Numerical study of roundness effect on flow around a circular cylinder. *Physics of Fluids*, *32*(4). https://doi.org/10.1063/5.0002997

Wang, Y., Raja, V., Madasamy, S. K., Padmanaban, S., AL-bonsrulah, H. A. Z., Ramaiah, M., Rajendran, P., Raji, A. P., Muzirafuti, A., & Wang, F. (2022). Multi-Parametric Investigations on Aerodynamic Force, Aeroacoustic, and Engine Energy Utilizations Based Development of Intercity Bus Associates with Various Drag Reduction Techniques through Advanced Engineering Approaches. *Sustainability*, *14*(10), 5948. https://doi.org/10.3390/su14105948

Wei, Y., Tomkowski, R., & Archenti, A. (2020). Numerical Study of the Influence of Geometric Features of Dimple Texture on Hydrodynamic Pressure Generation. *Metals*, *10*(3), 361. https://doi.org/10.3390/met10030361

Ye, Q., Ou, Y., Xiang, G., & Chen, J. (2024). Numerical Study on the Influence of Water Depth on Air Layer Drag Reduction. *APPLIED SCIENCES-BASEL*, *14*(1). https://doi.org/10.3390/app14010431

Zhang, Z., Wang, Q., & Zhang, S. (2024). Review of Computational Fluid Dynamics Analysis in Biomimetic Applications for Underwater Vehicles. *Biomimetics*, *9*(2). https://doi.org/10.3390/biomimetics9020079

Gantt Chart for PSM 1																
No	Task Project	Plan / actual	Week													
			1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	PSM Title Registration	Plan	■													
		Actual	■													
2	Project Briefing	Plan		■												
		Actual		■												
3	Research About the Title	Plan		■	■	■										
		Actual		■	■	■										
4	Literature review (Chapter 2)	Plan			■	■	■	■								
		Actual			■	■	■	■								
5	Introduction (Chapter1)	Plan						■	■	■	■					
		Actual						■	■	■						
6	Methodology (Chapter 3)	Plan						■	■	■	■	■				
		Actual						■	■	■	■	■				
7	Choosing Method for The Project	Plan						■	■	■	■	■	■			
		Actual						■	■	■	■	■	■			
8	Benchmark Design	Plan								■	■	■	■	■		
		Actual								■	■	■	■	■		
9	Writing full report	Plan										■	■	■		
		Actual										■	■	■		
10	Submit draft to supervisor	Plan													■	
		Actual													■	
11	Project presentation	Plan														■
		Actual														■

APPENDIX B

No	Task Project	Plan / actual	Week 1	2	3	4	5	6	7	8	9	10	11	12	13	14
	Gantt Chart for PSM 2															
1	Benchmark Design/Meshing	Plan	▮	▮						▮						
		Actual	▮	▮						▮						
2	Run Analysis on Benchmark (3-18 million)	Plan	▮	▮						▮						
		Actual	▮	▮	▮					▮						
3	Modified Model Design	Plan	▮	▮	▮					▮						
		Actual	▮	▮	▮					▮						
4	Modified Model Meshing	Plan			▮	▮	▮			▮						
		Actual			▮	▮	▮			▮						
5	Report Writing Chapter 4	Plan		▮	▮	▮	▮	▮	▮	▮	▮					
		Actual		▮	▮	▮	▮	▮	▮	▮	▮					
6	Analysis on Modified Model	Plan			▮	▮	▮	▮	▮	▮	▮	▮				
		Actual			▮	▮	▮	▮	▮	▮	▮	▮				
7	Report Writing Chapter 5	Plan								▮		▮	▮			
		Actual								▮		▮	▮			
8	Finalize PSM 2 Report	Plan								▮				▮		
		Actual								▮				▮		
9	Submission of PSM 2 Report	Plan								▮					▮	
		Actual								▮					▮	
10	Presentation Preparation	Plan								▮					▮	
		Actual								▮					▮	
11	PSM 2 Presentation	Plan								▮						▮
		Actual								▮						▮

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (Continuity\ Equation)\ (1)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial u_i}{\partial x_j}\right) + f_i \quad (Momentum\ Equation)\ (2)$$

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu_t \frac{\partial k}{\partial x_j}\right) + P_k - \epsilon \quad (Turbulent\ Kinetic\ Energy)\ (3)$$

$$\frac{\partial \epsilon}{\partial t} + u_j \frac{\partial \epsilon}{\partial x_j} = \frac{\partial}{\partial xj}\left(\mu_t \frac{\partial \epsilon}{\partial xj}\right) + C_1 \frac{\epsilon}{k} P_k - C_2 \frac{\epsilon^2}{k} \quad (Dissipation\ Rate)\ (4)$$

APPENDIX C

Analsysis Result Using ANSYS FLUENT

Pressure
Contour 1

	[Pa]
2.490e+02	
1.968e+02	
1.446e+02	
9.237e+01	
4.016e+01	
-1.205e+01	
-6.426e+01	
-1.165e+02	
-1.687e+02	
-2.209e+02	
-2.731e+02	
-3.253e+02	
-3.775e+02	
-4.297e+02	
-4.819e+02	
-5.341e+02	
-5.863e+02	