

COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF PERODUA AXIA BODY WITH SEVERAL TYPES OF SPOILER



BACHELOR OF MECHANICAL ENGINEERING TECHNOLOGY (AUTOMOTIVE TECHNOLOGY) WITH HONOURS



Faculty of Mechanical and Manufacturing Engineering Technology



Wan Muhammad 'Afif Bin Wan Azir

Bachelor of Mechanical Engineering Technology (Automotive Technology) with Honours

COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF PERODUA AXIA BODY WITH SEVERAL TYPES OF SPOILER

WAN MUHAMMAD 'AFIF BIN WAN AZIR



Faculty of Mechanical and Manufacturing Engineering Technology

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

DECLARATION

I declare that this Choose an item. entitled "Computational Fluid Dynamics Analysis of Perodua Axia Body with several types of spoiler." 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.



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 Technology) with Honours.

Dr Setyamartana Parman Senior Lecturer Signature : ical & Manufacturing Engineering Tecl DON Universiti Teknikal Malaysia Melaka Supervisor Name DR. SETYAMARTANA PARMAN Date 18/1/2022 **UNIVERSITI TEKNIKAL MALAYSIA MELAKA**

DEDICATION

I dedicate my dissertation work to my family and many friends. A special feeling of gratitude to my loving parents, Wan Azir and Hamidah. My siblings Wan Adib, Wan Adibah, Wan Afiq, Wan Atiq and Wan Atif.

I also dedicate this dissertation to many friends who have supported me throughout the



ABSTRACT

This thesis presents the computational fluid dynamics analysis of the Perodua Axia body with several types of spoiler. The design of the Perodua Axia body is was created with several types of spoiler to analyze the drag force at the rear back. A spoiler's design purpose is to minimize drag and improve fuel economy. From the rear border of the roof to the trunk or tail of the automobile, many automobiles have a rather steep downward inclination. Flow separation occurs when air moving across the roof tumbles over this edge at a higher velocity. The airflow becomes turbulent, resulting in the formation of a low-pressure zone, which increases drag. Adding a spoiler to the vehicle's very back reduces flow separation by making the air slice longer and softer from the roof to the spoiler. Reduced flow separation reduces drag, which improves fuel efficiency; it also aids in keeping the back window clear by allowing air to flow easily through it. Due to the constraints of traditional wind tunnel experiments and quick advances in computer power, much effort has been devoted to studying vehicle aerodynamics computationally during the last decade. A 3D computer model of the Perodua Axia car will be designed with the software Catia Autocad and be used as the base model. The other model will be designed with another spoiler design. The simulation will be run to determine the aerodynamic effects of the spoiler.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

ABSTRAK

Tesis ini membentangkan analisis dinamik bendalir pengiraan badan Perodua Axia dengan beberapa jenis spoiler. Reka bentuk badan Perodua Axia dicipta dengan beberapa jenis spoiler untuk menganalisis daya seret di bahagian belakang belakang. Reka bentuk badan perodua axia dibuat dengan dan tanpa spoiler untuk menganalisis daya tarik di bahagian belakang. Tujuan reka bentuk spoiler adalah untuk meminimumkan daya seret dan meningkatkan ekonomi bahan bakar. Dari sempadan belakang bumbung ke batang atau ekor kenderaan, banyak kereta mempunyai kecenderungan ke bawah yang agak curam. Pemisahan aliran berlaku apabila udara yang bergerak melintasi bumbung jatuh di tepi ini pada halaju yang lebih tinggi. Aliran udara menjadi bergelora, menghasilkan pembentukan zon tekanan rendah, yang meningkatkan daya tarik. Menambah spoiler di bahagian belakang kenderaan akan mengurangkan pemisahan aliran dengan menjadikan hirisan udara lebih panjang dan lebih lembut dari bumbung ke spoiler. Pemisahan aliran yang dikurangkan mengurangkan seretan, yang meningkatkan kecekapan bahan bakar; ia juga membantu menjaga tingkap belakang tetap bersih dengan membiarkan udara mengalir dengan mudah melaluinya. Oleh kerana kekangan eksperimen terowong angin tradisional dan kemajuan cepat dalam daya komputer, banyak usaha telah dikhaskan untuk mempelajari aerodinamik kenderaan secara komputasi selama dekad yang lalu. Model komputer 3D kereta perodua axia akan dirancang dengan perisian Catia Autocad dan digunakan sebagai model asas. Model lain akan direka dengan reka bentuk spoiler lain. Simulasi akan dijalankan untuk menentukan kesan aerodinamik spoiler.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

ملسيا ملا

ACKNOWLEDGEMENTS

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

First and foremost, I would like to thank and praise Allah the Almighty, my Creator, my Sustainer, for everything I received since the beginning of my life. I would like to extend my appreciation to the Universiti Teknikal Malaysia Melaka (UTeM) for providing the research platform.

My utmost appreciation goes to my main supervisor, DR. SETYAMARTANA PARMAN, from the Faculty of Mechanical and Manufacturing Engineering Technology, Universiti Teknikal Malaysia Melaka for all his support, advice, and inspiration. His constant patience for guiding and providing priceless insights will forever be remembered.

Finally, I'd want to express my gratitude to Universiti Teknikal Malaysia Melaka (UTeM) for providing me with the chance to study and develop experience during the last three



TABLE OF CONTENTS

		PAGE
DEC	LARATION	
APP	ROVAL	
DED	ICATION	
ABS	ГКАСТ	i
ABS	ГКАК	ii
АСК	NOWLEDGEMENTS	iii
TAB	LE OF CONTENTS	iv
LIST	OF TABLES	vii
LIST	OF FIGURES	viii
LIST	OF SYMBOLS AND ABBREVIATIONS	x
LIST	OF APPENDICES	vii
		лп
	PTER 1 INTRODUCTION	13
1.1	Background	15
1.2	Research Objective TI TEKNIKAL MALAYSIA MELAKA	14 15
1.4	Scope of Studies	16
СНА	PTER 2 LITERATURE REVIEW	17
2.1	Introduction	17
2.2	Automotive Aerodynamics Technology's History	17
	2.2.1 Passenger Car	19
1 2	2.2.2 City Car	23
2.3	2.3.1 Computational Fluid Dynamics	24 24
	2.3.2 Aerodynamics	24
	2.3.3 Problem of Aerodynamics Classes	25
	2.3.4 Aerodynamics of Automotive	25
2.4	Spoiler	26
2.5	Structural Concept of An Adaptive Shock Control Bump Spoiler	27
2.6	Theory of Aerodynamics	31
	2.6.1 Theorem of Bernoulli	31
	2.6.2 Aerodynamic Drag	32
	2.6.3 Aerodynamic Lift	33
	2.0.4 Development of the Boundary Layer	34

2.7	Flow Separation on Vehicle	35
	2.7.1 Introduction	35
	2.7.2 Process of Flow Separation	35
	2.7.3 External Flow	36
2.8	General Features of Fluid Flow	38
	2.8.1 Laminar and Turbulent Flow	38
	2.8.2 Flow Attached and Flow Separated	39
2.9	Properties of Fluids	40
	2.9.2 Pressure	41
	2.9.3 Temperature	41
	2.9.4 Viscosity	41
	2.9.5 Specific Heat	42
	2.9.6 Coefficient of Heat Transfer, k	43
СНА	PTER 3 METHODOLOGY	44
3.1	Introduction	44
3.2	General Methodology	44
	3.2.1 Proposed Methodology	45
3.3	Data Research and Analysis	46
	3.3.1 Instrument of Research	46
	3.3.2 Design Research	46
3.4	Design of Perodua Axia	47
3.5	CATIA	49
3.6	Ansys Software	50
3.7	Simulation of Car Body	51
	3.7.1 Geometry and domain of car	51
	3.7.2 Meshing	52
	3.7.3 Solver Setting	53
	3.7.4 Residual Monitors	54
3.8	Summary of this chapter	54
CHA	PTER 4 RESULTS AND DISCUSSION	55
4.1	Introduction	55
4.2	Results	56
	4.2.1 Velocity Streamlines	56
	4.2.2 Static Pressure	57
1 2	4.2.3 Velocity Contour	58
4.3	Discussion	60
	4.3.1 Drag Coefficient	60
1 1	4.5.2 Lift Coefficient	02
4.4	Chapter Summary	64
CHA	PTER 5 CONCLUSION AND RECOMMENDATION	65
5.1	Conclusion	65
5.2	Recommendations	66
REF	ERENCES	67



vi

LIST OF TABLES

TABLE	TITLE	PAGE
Table 2.1: Typical passenger ca	r drag and lift coefficients	20
Table 2.2: On a conventional se	dan, a computed breakdown of the drag components	20
Table 2.3: Geometric defnitions	s for the investigation of a position variable SCB	27
Table 3.1: Material Properties		51
Table 3.2: Boundary Conditions	\$	51
Table 3.3: Solution Methods		52
Table 4.1: Drag Coefficient of c	car body	60
Table 4.2: Drag Force on Car Bo	ody	60
Table 4.3: Lift Coefficient of Ca	ar Body	62
Table 4.4: Lift Force on Car Boo	اونيومرسيتي تيكنيكا يه	62
UNIVERSITI T	EKNIKAL MALAYSIA MELAKA	

LIST OF FIGURES

FIGURE TITLE	PAGE
Figure 2.1. Vehicle dynamics in commercial vehicles throughout time	14
Figure 2.2: Passenger automobile designs that are most commonly found	16
Figure 2.3: Comfort-enhancing aerodynamic gadgets include: A different r	ear wind
deflector located	18
Figure 2.4: Venturi phenomenon	21
Figure 2.5: Flow through the plate	24
Figure 2.5: SCB geometry definition	27
Figure 2.6: Structural concept with pressurized cells for realization of a sho	ock control
bump spoiler	28
Figure 2.7: Structural concept for a shock control bump spoiler by preshapi	ing 29
Figure 2.8: Sketch of the adaptive SCB-Spoiler (reference) concept with m	ain
contributing elements	29
UNIVERSITI TEKNIKAL MALAYSIA MELAKA Figure 2.9: Venturi phenomenon	A 30
Figure2.10: Flow through the plate	33
Figure 2.11: Process of flow separation	35
Figure 2.12: Boundary layer	36
Figure 2.13: Flow on a vehicle	37
Figure 2.14 Flow Diagram: a) laminar flow b) turbulent flow	37
Figure 2.15 Flow diagram: a) Attached flow B) Separated flow	38
Figure 3.1: Project methodology flowchart	44
Figure 3.2: Perodua Axia drawing with spoiler	46
Figure 3.3: Perodua Axia drawing without spoiler	46

Figure 3.4: The measurement of Perodua Axia body that used in CATIA	47
Figure 3.5: Spoiler A	48
Figure 3.6: Spoiler B	48
Figure 3.7: Geometry of Perodua Axia Car	50
Figure 3.8: Domain of the Geometry	50
Figure 3.9: Meshing of Car Profile	51
Figure 3.10: Solver Settings	52
Figure 3.11: Residual Monitors	53
Figure 4.1: Velocity streamlines around car body with spoiler A	56
Figure 4.2: Velocity streamlines around car body with spoiler B	57
Figure 4.3: Static Pressure on Perodua Axia with Spoiler A	58
Figure 4.4: Static Pressure on Perodua Axia with Spoiler B	58
Figure 4.5: Velocity Contour on Perodua Axia with Spoiler A	59
Figure 4.6 Velocity Contour on Perodua Axia with Spoiler B	59
Figure 4.7: Drag Coefficient of Car Body	61
Figure 4.8: Drag Force on Car Body	61
Figure 4.9: Lift Coefficient of Car Body KAL MALAYSIA MELAKA	63
Figure 4.10: Lift Force on Car Body	63

LIST OF SYMBOLS AND ABBREVIATIONS

Р	- Pressure
Р	- Density
V	- Velocity
F _d	- Force Drag
C _d	- Drag Coefficient
А	- Area
F _L	- Force Lift
C _L	- Lift Coefficient
Re	- Reynolds number
L	Length
μ	- Viscosity of the fluid
m	اويوم سيني بيڪيڪل مليسيا مارڪ
V	UNIVEVOLUME TEKNIKAL MALAYSIA MELAKA
F	- Force
u/y	- Rate of shear deformation
Q	- Heat energy
с	- Specific heat capacity
ΔT	- Change in temperature
h	- Heat transfer coefficient
q	- Heat flux
ΔT	- Difference in temperature between a solid surface and surrounding
	fluid



xi

LIST OF APPENDICES

APPENDIX

TITLE

PAGE

70

Appendix A: Orthographic View

UTERSITI TEKNIKAL MALAYSIA MELAKA

CHAPTER 1

INTRODUCTION

1.1 Background

Aerodynamics is the study of how things flow through the air. Aerodynamics is a field of fluid dynamics that examines the motion of air (thomas,2016). The laws of aerodynamics describe how an object interacts with air in motion. Aerodynamics effect anything that moves freely, from a jet to a banner. Because they are accompanied by air, even automobiles are impacted by aerodynamics.

The body and chassis of the vehicle can be built to reduce total weight and enhance overall aerodynamic characteristics for better riding circumstances, higher navigational accuracy, and reduced energy usage. More power under the hood means higher rotational speeds can be produced, for which become loses its aerodynamic properties are insufficient to provide the necessary downforce and handling. The aerodynamic properties of an automobile have a major impact on its performance, handling, protection, and comfort. Additional body components, such as rear spoilers, lowered front and rear skirts, airflow dams, and other aerodynamic assists, are installed to guide airflow in a new direction and reduce drag while improving stability.

Owners of daily vehicles nowadays tend to change their vehicles to make them look more sporty (Katz, 2016). A rear spoiler is a part that aids in the generation of downforce in a car. It's an aerodynamic device that "ruins" adverse airflow across a vehicle's body. As an air barrier, a spoiler could be installed on the front or back bumper. Spoilers can be attached to any part of a car, although the most common location is in the back. This is also determined by the form of the car's rear end, which might be rectangular, notchback, or sportys sedan. Lift and drag are two main aerodynamics factors that the rear spoiler contributed to. The spoiler can also be utilised to control cornering balance and conserve power by reducing drag force. A rear spoiler can reduce drag and back wheel boost while somehow decreasing dust on the back floor.

1.2 Problem Statement

The automobile has a strong tendency to lift over when driven at a fast speed, particularly on a road with a speed of traffic of 110 km/h. The pressure decreases as the higher amount of air in front of the glass accelerates as it passes over the glass. This decreased pressure elevates the roof as the air travels over the car. When the air is expelled, the notch created by the window falling to the body creates a vacuum or low pressure area that the air cannot adequately cover.

The wind is stated to split, and the consequent lower pressure creates lift, which affects the body's surface area. To reduce lift and increase high pressure, a rear spoiler can be added to the vehicle's body. Spoilers are most commonly found on sedans. They act as airflow barriers, allowing for increased air pressure at the front of the spoiler. This is beneficial because, as stated earlier, the low-pressure zone above the trunk lifts the car's back end, making it look "Light." At high velocities, its key goal is to decrease drag and air noise, lower noise emissions, and eliminate undesired lift forces and other causes of aerodynamic instability. The air is likewise considered a fluid in this circumstance.. The chief purpose of a spoilerare to make the car looking awesome. It is also to improve the downforce for a car that traels at high speed. Aerodynamic drag increases as a vehicle speeds up, making the engine work harder to maintain speed. Additionally, more air gets beneath it, causing "lift," which weakens grip and makes the car less stable. The quantity of air that passes beneath the car is reduced by front spoilers. Spoilers produce greater downforce towards the back, where airflow is more turbulent and generates more lift, keeping the car firmly planted on the road. So when the drivers are interested on installing the spoiler, the drivers cannot decide which are the best option for the car. Perodua Axia car was choosed to be analyzed with several types of spoiler. It is because Perodua Axia is well-known for being the entry-level car. Its also comes with affordable price and small dimensions, which is the best option as the perfect starter car for a new driver. So the whole body of Axia will be selected to be analyzed using CATIA and Ansys, it will be tested with several types of spoiler.

1.3 Research Objective TEKNIKAL MALAYSIA MELAKA

کنکا ملیسہ

- a) To study aerodynamic performance of city car that is Perodua Axia by Computational Fluid Dynamics method.
- b) To investigate the impact of the several type of spoiler on the aerodynamic performance of Perodua Axia by the Computational Fluid Dynamics method.

1.4 Scope of Studies

The following are the parameters of the this study:

- Design of Perodua Axia body performed by using CATIA.
- Perodua Axia's design is used as a comparison for the aerodynamics performance.
- Ansys system is use to perform Computational Fluid Dynamics analysis which is shows which spoiler are better when it comes to aerodyanmics.
- Simulation of pressure and airflow through a car and the impact of the rear spoiler are the main focus of this project.



CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Automobiles can travel at such a high velocity rate that they can lose balance and cause accidental injuries as a result of their accelerated speed. This demonstrates the need for a controlling element such as aerodynamic wing with a spoiler that produces a carefully balanced body to stall over the wing section behind the spoiler, minimizing the lift of that wing section (Remer, 2019). Spoilers are engineered to minimize lift while also raising drag significantly.

As an air dam, a spoiler may be added to the front or rear bumper. The rear spoiler contributed to two major aerodynamic factors: lift and drag. The spoiler can also be utilized to control cornering stability and conserve power by reducing drag force. A rear spoiler can help minimize drag and back wheel boost, as well as dust on the rear ground.

2.2 Automotive Aerodynamics Technology's History

The merging of aerodynamics with automotive technology has taken a long time. Only after numerous efforts has a synthesis of the two proved effective (Thomas,2016). This is unexpected, because collaboration with fluid mechanics has proven to be quite beneficial in the surrounding fields of traffic technology, naval design, and aeronautics. Of course, ship and airplane designers had a leg up. They got their inspiration from fish and birds in nature. Many fundamental characteristics were taken from these natural forms. In the automobile, there were no such originals. As a result, its architects attempted to mimic the forms of ships and airplanes, which must have seemed sophisticated to them at the time. This rapidly proved to be the incorrect technique. Aerodynamics made a breakthrough in the car only when it broke free from these faulty originals.

Some other reason for early and repeated vehicle aerodynamic problems is that they began much too early. The early vehicles were sluggish and inefficient. Streamlined bodies would have looked absurd on the poor roads of the day. The original design of horse-drawn carriages protected the people in the car from wind, mud, and rain. Eventually, the stereotype that aerodynamic bodies are just for peculiar individuals won over the need to gain from aerodynamics for economic reasons.

Figure 2.1 depicts a brief history of vehicle aerodynamics. People, the majority of whom were not from the motor business, worked on aerodynamic development throughout the first two of the four eras. They attempted to apply fundamental ideas from aviation aerodynamics to automobiles. Later, throughout the last two periods, automobile firms took up the discipline of vehicle aerodynamics and integrated it into product development. Aerodynamics has been (and continues to be) the responsibility of teams rather than individual inventors since then.

UNIVERSI	Basic shapes	1900 to 1925	Torpedo Boat t	ail Air ship	MELAKA
		1921 to 1923	Rumpler	Bugatti	
	ined cars	1922 to 1939	Jan	av av	
	Stream	1934 to 1939	Kamm	Schlör	
		Since 1955	Citröen	NSU-Ro 80	
	Detail optimization	Since 1974	VW-Scirocco I	VW-Golf I	
	Shape optimization	Since 1983	Audi 100 III	Ford Sierra	

Figure 2.1: Vehicle dynamics in commercial vehicles throughout time.

(Thomas, 2016)

A torpedo-shaped vehicle with a small drag coefficient was the first to be developed utilizing aerodynamic principles, but the revealed passenger being out of body axles should have disrupted the vehicle's favorable flow characteristics (Thomas,2016). In comparison to planes and submarines flying in a medium that encloses the body, they were unconcerned with the body's proximity to the ground. The available wheels and exposed undercarriage, paired with the ground, interrupt the flow of an automobile like this.

As time passes, more research into the effects of aerodynamics on automobiles is conducted, and new designs are produced to meet growing demands and to save money. The wheels evolved to be built into the body, decreasing aerodynamic drag and resulting in a smoother flow. To keep the streamlining attached for many years, the tail was long and oddly shaped. Smooth bodywork, connected bumpers, and body-integrated headlamps were all added to the cars. The designers had designed a car with a unique shape from traditional horse-drawn vehicles. Researchers had unquestionably managed to develop cars with low drag coefficients.

2.2.1 Passenger Car

One of the general forms illustrated in figure 2.2 might be one of the possible versions supplied by a specific manufacturer (K. Joseph,2016). The measuring methods and facilities utilized usually have an impact on the given aerodynamic data. For example, most companies will do standard car testing on the street or in a wind tunnel. Whether or not moving ground or ambient variables are employed in coast down testing, however, may have an impact on the results. A SUV has significantly lesser drag than a sedan or a well-equipped hatchback.

Furthermore, open-top automobiles (sedans) have a flow split below the window, which explains why they have a larger drag. SUVs are based on modern automobiles and feature a rectangular form with sharp points, resulting in the highest drag (K. Joseph,2016). Furthermore, it is well accepted that driving with windows closed and open-air con saves gasoline. since the drag of a vehicle increases when the windows are opened. While considering a car with totally closed windows to one with fully open windows, Table 2.1 provides average incremental drag coefficient results. The most significant increase is seen in boxy forms, such as the SUV depicted. Furthermore, opening merely single window at moderate rates creates lower frequencies pressure fluctuations (buffeting), that might very annoying. Table 2.1 shows the typical progressive drag coefficient value evaluating in the vehicle with totally closed or fully opened windows.



Figure 2.2: Passenger automobile designs that are most commonly found (K.

Joseph,2016).

	C _D	C _{Lf}	C _{Lr}	ΔC_D (open window)
Sedan	0.32	0.067	0.114	~0.05
Wagon	0.30			~0.04
Hatchback	0.31			~0.03
Convertible	0.40	0.011	0.143	
SUV	0.40-0.50			~0.06

Table 2.1: Typical passenger car drag and lift coefficients (K. Joseph, 2016).

It's also fascinating to see which parts of the car contribute to total drag (and how much). This is a challenging question to answer since a complete drag split is hard to quantify physically and may vary depending on the CFD method used. Some approximated amounts referred on calculations are showed in table 2.2.

3	
	ΔC_D
Bodywork	0.050
Rearview mirrors	0.015
shi ala isa	Si in sint
Rear surfaces	0.085
Engine bay UNIVERSITI TEKNIKAL	0.024.AYSIA MELAKA
Cooling drag	0.048
Underbody + chassis	0.085
F wheel + suspension	0.025
R wheel + suspension	0.023
Total drag coefficient	0.355

 Table 2.2: On a conventional sedan, a computed breakdown of the drag components

(K. Joseph, 2016).

In open-top cars, aerodynamics is frequently used to increase comfort. Even at modest speeds, the aerodynamic turbulence (pressure variations) generated by open a sedan's glass or sunroof can be uncomfortable. As an example, Figure 2.3a depicts the reversed flow behind the glass. The unequal reverse flow in this scenario might blast the driver's hair into his or her eyes, impeding focus, or just blow stuff surrounding the car away. A typical method for restricting the reverse flow channel is to employ a movable screen or a rear air deflector (see Figure 2.3b). These devices may be operated automatically, rising and retracting at high and low rates, respectively. As shown in Figure 2.3c, example like the wind deflectors installed on the top of the glass. Strong winds are reduced by directing the flow over the vehicle's entire open top. At low speeds, this approach is easy and effective, but at greater speeds, it increases drag.



Figure 2.3: Different wind deflector location: a) Without wind deflector b) Rear wind deflector c) Front wind deflector (K. Joseph,2016).

2.2.2 City Car

When it comes to the sort of car people drive, bigger isn't necessarily better. While some people—families, businesses that transport packages—need a large car or SUV, but are not ideal for everyone (W. Liu,2017). A tiny, little automobile is often all that is required to go around in daily life. Large automobiles and strong SUVs are far more difficult to manage than small cars. This is especially advantageous for individuals who live in congested cities or on narrow streets. Parallel parking, as well as squeezing into slots in a packed parking lot, are also simpler.

Compact automobiles are also simpler to maneuver in tight spaces. Smaller automobiles are more fuel efficient than larger vehicles. This translates to fewer pollutants harming our precious ecosystem. Small automobiles are made up of fewer materials, using less fuel to manufacture than bigger vehicles. A compact automobile will take up less room in a landfill after it is no longer drivable. Unless driver is looking for a high-end sports car, little automobiles are often less expensive than bigger ones.

Compact automobiles are less expensive to construct than bigger cars because they use less materials. This keeps the price down. Because of their cheaper pricing, compact automobiles gained appeal after the 2008 crisis (Steg, L,2005). Smaller automobiles require less maintenance than V6 or V8 vehicles since their engines are smaller, generally with four cylinders. 4-cylinder engines seldom require the belt change that occurs at 100,000 miles, which is inconvenient. Instead, they're usually driven by a chain, saving consumers a few thousand dollars when your car hits 100,000 miles.

2.3 Fluid Dynamics

In physics and engineering, fluid dynamics is a subcategory of fluid mechanics that describes the movement of liquids and gases (Mustafa Cakir, 2012). Two of its subfields are aerodynamics (the study of air and other gases in motion) and hydrodynamics (the study of water in motion) (the study of liquids in motion). Fluid dynamics is used to measure forces and moments on aeroplanes, quantify the mass flow rate of petroleum via pipelines, predict weather patterns, study nebulae in interstellar space, and estimate fission bomb blasts, to name a few applications.

2.3.1 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is a subset of computer science that incorporates computer science, mathematics, and modern fluid dynamics (Mustafa Cakir, 2012). CFD analyses and solves fluid flow problems using numerical methods. CFD is a tool for researching and analysing a wide variety of heat transfer and fluid flow issues. Automotive aerodynamics was among the most useful concepts of CFD.

2.3.2 Aerodynamics SITI TEKNIKAL MALAYSIA MELAKA

Aerodynamics is the science of how air flows through objects. The laws of aerodynamics describe how an aeroplane should fly (Kumar,2015). Aerodynamics affects anything that travels through the air. Aerodynamics affect both a rocket bursting from its launch pad and a kite flying through the air. Aerodynamics has an impact on cars since wind passes through them. The research of the aerodynamics of passenger cars is known as automotive aerodynamics. At full speed, the main objectives are to reduce drag and wind noise, as well as to avoid unwanted lift forces and other sources of aerodynamic instability. Air is frequently referred to as a fluid. For certain racing vehicle classes, producing downforce might be necessary to increase traction and thus cornering skills.

2.3.3 Problem of Aerodynamics Classes

Aerodynamics are an issue in many classes. External aerodynamics is the study of flow surrounding physical objects of various shapes and sizes, like studying lift and drag through a plane or shocks wave front of a rocket's nose (Mustafa Cakir, 2012). Internal aerodynamics, on the other hand, is the analysis of flow through solid matter such as a gas turbine or an air conditioners pipe through channels.

The ratio of typical flow speed to sound speed is the second type of aerodynamic categorization. If all of the speeds in an issue are less than the speed of sound, it is termed subsonic; if both below and above the speed of sound are present, it is called transonic. It is considered supersonic when the usual flow speed surpasses the speed of sound, and it is called hypersonic when the flow speed exceeds the speed of sound. The impact of viscosity is addressed in the third classification. Because certain problems have low viscous effects, the viscous might be deemed non-existent. Inviscid flows are the result of this issue, whereas viscous flows are the result of Viscosity.

2.3.4 Aerodynamics of Automotive

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

20

Automotive aerodynamics is the study of the aerodynamics of road vehicles. At high speeds, the objective of this study is to decrease drag, wind noise, and undesired lift forces (Mustafa Cakir, 2012). It's also important in various racing car classes to create the necessary downward aerodynamic force to improve traction and handling ability.

The wheel and lights will be incorporated into the aerodynamic vehicle to save space. It features a fastback-style tail and lacks a form edge that crosses the wind stream to be streamlined. To create desirable downward forces, the aerodynamic design will be flat and smooth.

2.4 Spoiler

A spoiler is an aerodynamic device for vehicles that is designed to "ruin" adverse airflow over the body of a moving vehicle, often known as drag. Air dams are front-end spoilers that, in addition to directing airflow, reduce the amount of air flowing beneath the vehicle, minimizing aerodynamic lift and drag. Spoilers are ubiquitous on race cars and high-performance luxury automobiles, but they've gained favor in normal vehicles as well. Spoilers on vehicles are generally employed for aesthetic reasons and give little or no aerodynamic benefit.

Many passenger car spoilers are designed to reduce drag and enhance fuel efficiency. For passenger automobiles, front and rear spoilers are offered. Front spoilers, which are located behind the fender, are generally aimed to decrease drag and enhance height by limiting the quantity of air travelling beneath the vehicle. In sports vehicles, rear and front spoilers are highly popular. Despite the fact that these cars often have a more compact chassis and better suspension to help in high-speed agility, a spoiler may be advantageous. Airflow separation can occur because certain vehicles have an extremely steep descending angle from the rear border of the top to the body or rear of the vehicle. The air flow becomes unstable, causing a low-pressure environment to form, which increases drag and turbulence. The installation of a rear spoiler allows the air to "see" a wider, softer slope from the roof to the spoiler, reducing flow isolation and decreasing lift on the car by producing down power.

2.5 Structural Concept of An Adaptive Shock Control Bump Spoiler

Drag reduction methods in aircraft design are a critical enabler for decreasing emissions and ensuring commercial aviation's long-term growth (Markus,2019). Drag reduction promises to be a significant benefit of laminar wing technology. The usage of shock control bumps is one approach of reducing the wave drag induced by the aerodynamic shock (SCBs). SCBs have the goal of converting a single powerful shock into multiple smaller and weaker -shocks, resulting in a drag benefit when placed appropriately. SCBs should be adjustable in position and height as the shock position changes with various parameters such as speed, altitude, and attack angle throughout the conflict. SCBs, on the other hand, can assist manage laminar buffeting as a fixed case by fixing the shock into certain places at the SCB's location. An adaptive shock control bump spoiler is proposed in this study as a structural idea. An adaptive spoiler with two conventional actuators is given, based on a fixed bump SCB spoiler idea. The interdependencies of key design factors are examined, as well as the design motivations.



Figure 2.4: Aerodynamic concept of shock control bumps (Sommerer, 2019)

Sommerer and colleagues presented computational results for adaptive transonic airfoils with variable camber and shock bump control. The findings indicate that the usage of SCBs on a transonic airfoil has a bright future. Sommerer enhanced bump shapes and investigated the impact of several geometric bump representations for a specific out-ofdesign scenario. The precise shape of the bump is demonstrated to be of minor importance, but bump location, or the positioning of the bump crest, and height have a substantial impact on bump efficacy. To obtain the greatest possible drag reduction, the structural design of a shock control bump should be able to change the bump height and placement to out-of-design and flow conditions. Based on these findings, a generic geometry is used to construct the position-adaptable SCB spoiler concept, with SCB design principles and suggestions provided in Table 1 and Fig. 2.



Table 2.3: Geometric defnitions for the investigation of a position variable SCB



Figure 2.5: SCB geometry definition

The author proposes an integrated design concept (Sousa,2015). A spoiler body with pressurized cells is built for an adjustable SCB (Fig. 3). The cells are integrated within the spoiler body and may create a variety of SCB shapes depending on their pressurization. The design is built on analytical methods enabling a speedy optimization procedure. As a preliminary design approach for the provided notion, the creation of target shapes with consideration of mechanical boundary conditions is offered.



Figure 2.6: Structural concept with pressurized cells for realization of a shock control bump spoiler (Sousa,2015)

The position-adaptive SCB spoiler idea is based on the fixed SCB concept proposed by Kirn,2016 and Machunze,2000. The concept is improved by the addition of a second actuator, which controls a second bump point. The fundamental notion of a pre-shaped spoiler body is kept, but it has been enlarged to incorporate a maximum backward and forward bump position. An adaptive SCB can be enabled and positioned between these points based on the flow conditions. A second hinge site at the spoiler's trailing edge is provided by the end of a stiffened section of the spoiler body. The bump location may then be adjusted depending on stiffness tailoring and actuator displacements in the region between the first and second hinge chord positions.



Figure 2.7: Structural concept for a shock control bump spoiler by preshaping

(Kirn,2016)



Figure 2.8: Sketch of the adaptive SCB-Spoiler (reference) concept with main contributing elements (Machnunze,2000)

The 'flexible hinge' approach of connecting the spoiler to the wing box proposed by Kirn,2016 is approved because the position-adaptive SCB design suggestion focuses especially on shock control for wings with natural laminar flow. However, because the flex-hinge technology is separate from the SCB technology, the author just utilises it as a design constraint and does not go into detail about it. To seal the fap gap, the spoiler must be in contact with the fap body when cruising. The trailing edge of the spoiler must establish a line contact with the fap with a particular contact force.
2.6 Theory of Aerodynamics

2.6.1 Theorem of Bernoulli

If a fluid flows horizontally with no change in gravitational potential energy, a decrease in fluid pressure equals an increase in fluid velocity, according to Bernoulli's theorem (Diakakis,2014). When a fluid flows through a straight tube with varying pass area, it accelerates in constrained sections and exerts the least amount of pressure when the cross-sectional area is the smallest. Figure 2.4 depicts the venturi effect phenomenon.



Bernoulli's principle, which culminates in Bernoulli's equation, may be utilized to many different types flow of fluid (Eq. 2.1). In reality, various types of flow need distinct versions of Bernoulli's equation. Bernoulli's principle in its simplest version is true for incompressible flows (most liquids) as well as compressible flows (gases).

$$P + \frac{1}{2}pv^2 = constant$$
 (2.1)

Where **P** = pressure

P = density

v = velocity

The principle of Bernoulli is the same as the idea of energy conservation. The sum of all kinds of mechanical energy in a fluid along a streamline is the same at all places along that streamline in a steady flow (Diakakis,2014). This implies keeping the total of kinetic and potential energy constant. Because the energy per unit mass (the sum of pressure and gravitational potential) remains the same throughout a reservoir, the sum of all forms of energy on all streamlines as the fluid flows out of it is the same.

2.6.2 Aerodynamic Drag

The force that keeps an item from moving through a fluid is known as drag. Aerodynamic drag arises when the fluid is a gas, such as air. Hydrodynamic drag arises when the fluid is a liquid, such as water (Diakakis,2014). Drag is a complex phenomenon that is exceedingly difficult to describe from a hypothesis based only on fundamental principles.

The capacity to flow is what distinguishes fluids. A fluid, in semi-technical terms, is any substance that cannot withstand a shear force for an extended period of time. This makes them tough to grip while making pouring, stirring, mixing, and distributing them simple. Consequently, fluids take on the form of their container rather of having a distinct shape. Fluids are unique in that, when contrasted to solids, they give up their space rather easily to other material objects.

Fluids aren't solids, yet they are definitely stuff. The ability to have both mass and volume is a fundamental feature of materiality. Changes in velocity are resistant to material

items, and no two material objects may be at the similar position around the same moment. The pressure drag is the fraction of the drag force caused by the fluid's inertia, or its urge to shift when it is pushed aside to make room for something else.

$$C_{d} = \frac{Fd}{\frac{1}{2}pv2A}$$
(2.2)

Where F_d = Force of Drag

 $C_d = Drag of Coefficient$

ALAYS/A

p = Density

v = velocity

A = Area

2.6.3 Aerodynamic Lift

In passenger automobiles, lift is typically of little consequence since the speed is usually too low to create significant lift. Something unusual happened at high speeds, which was observed early on: the automobile seemed to rise off the ground. Lift may be dangerous, especially in race vehicles. It has a significant impact on the car's control and handling (Diakakis,2014).

The airflow over the top of an automobile is quicker than the airflow across the bottom, causing lift. This happens in all automobiles to some extent. Bernoulli's theorem states that as speed rises, pressure drops. As a result, the pressure at the roof of the vehicle is lower than at the base, resulting force lift.

The amount of lift generated by an object is influenced by the density of the air, the velocity between some of the object and the air, the viscosity and compressibility of the air, the surface area over which the air flows, the shape of the body, and the body's inclination to the flow, also known as the angle of attack.

$$C_{\rm L} = \frac{FL}{\frac{1}{2}pv2A} \tag{2.3}$$

Where F_L = Force Lift

 $C_L = Lift Coefficient$ p = Density v = velocityA = Area

2.6.4 Development of the Boundary Layer

The boundary layer can be defined as the fluid layer immediately around a bounding surface in physics and fluid mechanics (Diakakis,2014). There's always a velocity boundary layer and hence surface friction for flow through any surface. This surface friction on a car's body might cause drag, which could affect the car's aerodynamics.



Figure 2.10: Flow through the plate (Diakakis, 2014)

Boundary layer velocity is generated by viscous processes associated to relative motion between a fluid and a surface. A flow region is defined by the gradients of shear stresses (t) and velocity (u). As the flow direction changes, the thickness of the region between the surface and the free stream increases. Surface friction is heavily influenced by flow conditions, whether laminar or turbulent. As you go closer to the back of the automobile, the boundary layer on the surface thickens. The greater the thickness of the boundary layer, the more difficult it is for air to separate from the body, resulting in turbulent flow.

2.7 Flow Separation on Vehicle

2.7.1 Introduction

Understanding how air flows through a vehicle's body is critical for determining and analyzing its aerodynamics. In general, air flows around the body of a vehicle may be divided into two types: turbulent and laminar. Both layers may be characterized as the boundary layer grows on the car's surface.

2.7.2 Process of Flow Separation

Aside from turbulence, the boundary layer also has a phenomenon known as flow separation, which generates pressure drag on a body (see Figure 2.6). It happens when the pressure at the back surface is lower than the pressure at the front surface, causing separation and resulting in drag. This may be noticed while investigating flow from the leading edge to the following edge of an aero foil (Katz,2016).

Flow separation is unfavorable because it generates a larger wake and lower pressure on the back side, resulting in slower pressure restoration. To avoid insufficient flow separation, airflow movements from the roof to the rear window must be greatly simplified. The lack of separation may be a factor in the drag. Aerodynamics will be significantly more effective if the flows operate in clean air (laminar flow).



Figure 2.11: Process of flow separation (Katz, 2016)

2.7.3 External Flow

Figure 2.8 depicts the movement of air around a vehicle. The road speed of the automobile is the undisturbed velocity (V) in still air. If there is no flow split, the viscous effects in the fluid are restricted to a thin layer of a few millimeters thickness known as the boundary layer. Figure 2.7 demonstrates that the flow is inviscid beyond this layer, as well as its pressure is applied to the boundary layer (Katz,2016).

The velocity within the boundary layer reduces to zero at the wall, where the fluid fulfils the no-slip condition, from the value of the inviscid external flow at the boundary UNIVERSITY TEKNIKAL MALAY SIA MELAKA layer's outside border. The flow separates (Figure 2.8 shows separation immediately behind the back), the boundary layer is spread, and viscous processes completely control the flow.

When compared to the vehicle's typical length, such areas are relatively substantial. There is no change in velocity in between free stream and the ground at a given distance from the vehicle. For wind tunnel simulations of airflow over ground vehicles, this discovery is crucial.



Figure 2.12: Boundary layer (Katz,2016)



The Reynolds number is the name for this dimensionless parameter. As illustrated in Fig. 2.8, it is a function of the speed (V), the kinematic viscosity (v), and length (L). The kind of viscous flow around an item is influenced by its Reynolds number and form. Flows with differing Reynolds numbers for the same body form may have completely distinct properties. The Reynolds number is thus the dimensionless parameter that describes a viscous flow.



Figure 2.13: Flow on a vehicle (Katz,2016)

2.8 General Features of Fluid Flow

Aerodynamics is "limited" to the flow of air, whereas fluid dynamics is concerned with the motion of fluids (Mustafa Cakir, 2012). Fluids, unlike solids, cannot acquire a stable shape when loaded and will distort instantly. If we put a block in the backyard pool, for example, it will sink since the fluid beneath it is not stiff enough to support it.

2.8.1 Laminar and Turbulent Flow





Laminar flow is described as a fluid flow in which the fluid layers do not combine and move in opposite directions. In a laminar flow, the surface flow moves in a single direction. Laminar flow occurs when fluid flows at a low velocity in small diameter pipelines. A fluid flow with a Reynolds number less than 2000 is referred to as laminar flow. The fluid flow is strongly coupled in the sense that neighbouring layers of fluid do not mix and instead go straight toward one another and the tube wall. Shear stress in laminar flow is determined by fluid viscosity, which is unaffected by density (Thomas,2016).

Turbulent flow is described as a fluid flow in which adjacent layers of the fluid overlap one another instead of flowing in a straight line. The fluid layers in turbulent flow do not travel in a straight line. They move in a random zigzag pattern. When the velocity of the fluid is high and it flows through larger diameter pipes, the consequence is turbulent flow. A fluid flow with a Reynolds number greater than 4000 is considered turbulent. The fluid does not move in a logical order. Different strata are mixed together, and they do not move in a straight line, but rather cross each other. The density of a turbulent flow determines the shear stress (Thomas,2016).



Figure 2.15 Flow diagram: a) Attached flow B) Separated flow (Mustafa Cakir, 2012)

The flow that has not detached from the body is referred to as attached flow. In an airfoil, for example, the flow is connected at low angles of attack, i.e. the streamlines "stick" to the body. A separated flow is a liquid or gas flow when the flow around a body splits in its top, resulting in the formation of a reverse-vortex flow area. Small pressure gradients and low stagnation pressure values are typical of the reverse-vortex flow region.

Separated flow occurs in the border layer in particular. For the separated flow creation, a rise in pressure along the wall along the flow direction is required. In this circumstance, the particles travelling in the boundary layer near the body surface have a low velocity, therefore their kinetic energy is inadequate to counteract the increased pressure.

2.9 **Properties of Fluids**

ALAYSIA

Many properties related to thermodynamics, mechanics, and other fields of science may be present (Katz, 2016). We can only list a handful in the following paragraphs that are included in introductory fluid mechanics.

2.7.1 Density

By definition, density is defined as mass (m) per unit volume (Mustafa Cakir, 2012). More specifically its volumetric mass density that also know as specific mass. The density of a substance is usually measured in grammes per cubic centimetre. Density is defined as: اونيوم سيتي تيڪن<u>س</u>ڪل مليسيا ملاك

(2.5)

Where p = density ERSITI TEKNIKAL MALAYSIA MELAKA

m = mass

v = volume

2.9.2 Pressure

The pressure (P) is defined as the usual force (F) acting on a surface per unit area (A) (Mustafa Cakir, 2012). Pressure is a scalar quantity. It connects the normal force operating on the surface to the vector area element (a vector normal to the surface).

$$P = \frac{F}{A} \tag{2.6}$$

Where P = pressure

F = force

A = area

2.9.3 Temperature

Temperature is a measurement of hotness or coolness on one of several arbitrary scales that represents the natural direction of heat energy transfer (Mustafa Cakir, 2012). Temperature is not the same as the energy of a thermodynamic system. Temperature is classified as an intense characteristic, along with pressure and density.

2.9.4 Viscosity

The reluctance of a fluid (liquid or gas) to altering shape or movement adjacent portions relative to each other is referred to as viscosity. The resistance to flow is referred to as "viscosity." (Cakir,2012). The opposite of viscosity, which is a measure of flow efficiency, is defined as fluidity.

$$F = \mu A \frac{u}{v} \tag{2.7}$$

Where F = force

 μ = viscosity of fluid A = area u/y = deformation rate of shear

2.9.5 Specific Heat

The amount of heat required to increase the temperature of one gramme of a material by one Celsius degree is defined as specific heat (Cakir,2012). Specific heat is commonly expressed in calories or joules per gramme per degree Celsius. The amount of heat necessary to raise the temperature of a body by one degree in comparison to the amount required to raise the temperature of an identical quantity of water by one degree is referred to as specific heat.

 $O = mc\Delta T$

Where
$$Q = energy heat$$

(2.8)

m = mass VERSITI TEKNIKAL MALAYSIA MELAKA c = capacity of specific heat

 ΔT = temperature different

2.9.6 Coefficient of Heat Transfer, k

The coefficient of heat transfer is the amount of heat transmitted per kelvin per unit field (Mustafa Cakir, 2012). The coefficient of heat transfer (COPT) is a numerical representation of heat transfer coefficient in between fluid medium and the surface through which the fluid travels.

$$h = \frac{q}{\Delta T} \tag{2.9}$$

Where h = coefficient of heat transfer

q = flux of heat

 ΔT = temperature different



CHAPTER 3

METHODOLOGY

3.1 Introduction

The overall study process, procedure, or method employed in this project to fulfil the objectives is referred to as methodology. The methods followed throughout the project is discussed in this chapter. A flow chart is a diagram that depicts or explains a process. The project's overview, flow, and software development are discussed in this chapter.

3.2 General Methodology

In order to meet the project's goal, the following procedures are required:

a) Literature review

Reports, journal, article and books regarding the computational fluid dynamics will be reviewed. All the research has been reviewed in chapter 2 as knowledge and background of the aerodynamics and spoiler car.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

b) Drawing

Drawing of Perodua Axia body and spoiler will be draw in this project. There is two drawing of spoiler that been propose in this chapter.

c) Design

Design of Perodua Axia with several types of spoiler in Catia software. Two spoilers will be design in Catia software. Design will be analyzing in Ansys software to obtain results.

d) Creating reports

In light of the recently finished project, the report will be written as including the results and the discussion.

3.2.1 Proposed Methodology

In the flow chart below, a summary of the project methodology is shown.



Figure 3.1: Project methodology flowchart

3.3 Data Research and Analysis

A research and analysis tool are a tool that is used to gather, assess, and analyze data from study participants. It is necessary to choose an instrument based on the type of study that are conducting. Data analysis is the systematic application of statistical or logical tools to characterize, display, summarize, and interpret data.

3.3.1 Instrument of Research

The internet was used to find out more about spoilers, and web research time should be used to maximize efficiency while designing classes and activities. The steps of gathering as much information as possible include questioning, summarizing, and evaluating. Before going on the Internet, we should organize our questions to create a search strategy and a list of sites to investigate. The information is gathered by compiling a list of numerous questionnaires from which to select the most appropriate. Meeting with the project supervisor to discuss the project in further depth may be considered for a proper data collection.

3.3.2 Design Research ITI TEKNIKAL MALAYSIA MELAKA

This study uses an analysis approach to assess structural parameters such as internal geometry, behavior, and process. Spoiler design in terms of optimizing shape or having a substantial influence on spoiler lifespan necessitates the use of a basic dynamic model for analysis. Future study will concentrate on selecting the most appropriate bearing optimization parameter based on the bearing's design.

3.4 Design of Perodua Axia

3.3.1 Design A

Figure 3.2 shows the design of Perodua Axia body. This body are designed with the spoiler at rear back of car.



Figure 3.3: Perodua Axia drawing without spoiler

3.3.3 Dimension

Figure 3.12 show the drawing and mesurement for catia purpose. The unit are in milimeters(mm).



Figure 3.4: The measurement of Perodua Axia body that used in CATIA

3.3.4 Spoilers

Before designing a spoiler with the proper parameters, it must first refer to a genuine spoiler. The spoiler was created to show exactly which parameters need to be altered in order to decrease friction. But first, the spoiler must be measured in order to create the drawing. It may be redesigned in catia software by referring to both designs. All of the measurements were obtained and the brake pad was measured using a standard procedure. It is much easier to understand, diagnose, and propose solutions when a range of characteristics intended expressly for these reasons are employed.



abbreviation for Computer Aided Three-Dimensional Interactive Application (Amit,2016). It is more than a CAD (Computer Aided Design) application. It is a comprehensive software suite that combines CAD, CAE (Computer-Aided Engineering), and CAM (Computer-Aided Manufacturing) (Computer-Aided Manufacture). Computer-aided design is used by engineers to create, update, and/or analyse graphical representations of product designs. It has applications in a wide variety of areas due to its various popular characteristics. CAD software innovation is continuing to improve design quality by enhancing accuracy and reducing design errors. CAD software may assist enhance communication by providing engineers and manufacturers with a single source of truth due to the concentration of design data and documentation.

Engineers are the most likely to use computer-aided engineering to model and assess product designs. These sorts of research (MBD) include Finite Element Analysis (FEA), Computational Fluid Dynamics (CFD), and Multibody Dynamics (Amit,2016). These strategies, such as prototyping, help to prepare a product design for real-world issues. The findings of this testing may be used by engineers to make better product design decisions and upgrades, resulting in enhanced product performance and customer experiences. Manufacturers, not Design Engineers, are the most common users of computer-aided manufacturing for planning, organizing, controlling, and automating industrial processes. CAM software infers machining instructions from CAD designs, increasing the efficiency and usage of part production resources.

3.6 Ansys Software

Ansys Mechanical finite element analysis software is used to simulate computer models of structures, electronics, or machine components in order to evaluate their strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other properties (Leslie,2016). Ansys develops and sells engineering simulation software for use across the product life cycle. Static/dynamic, structural analysis, heat transport, and fluid difficulties, as well as acoustic and electromagnetic problems, are examples of these challenges.

3.7 Simulation of Car Body

Figure 3.7 below shows the design was created in CATIA software by referring the dimensions. After the creation of this geometry, it was imported to Ansys work bench design modeler in figure 3.8. In this design modeler, scaling has to be given to the profile in order to simulate analysis using CFD software.



3.7.1 Geometry and domain of car



Figure 3.8: Domain of the Geometry

3.7.2 Meshing



Figure 3.9: Meshing of Car Profile





Table 3.1: Material Properties

Inlet Condition (Velocity)	80km/hr
milet condition (velocity)	00Km/m.
Outlet Condition (Pressure)	$0 P_{2}$
Outlet Collution (Liessure)	014
Wall Condition	Stationary and no slip
	······································

Table 3.2: Boundary Conditions

Gradient	Least Squares Cell Based
Scheme	SIMPLE
Momentum	Second order up wind
Pressure	Standard

Table 3.3: Solution Methods

3.7.3 Solver Setting

The solver parameters to be specified in the solver are shown in Figure 3.10. The maximum number of iterations allowed in convergence control is 50. RMS denotes the residual type.

asic Settings	Equation Class Settings Advanced Options	
Advection Sche	me >	
option	High Resolution	-
Turbulence Nur	nerics	
Option	First Order	
Convergence C	Control	
in, Iterations Iax. Iterations Fluid Timescale	سيتي نيڪنيڪل آن د Control	اونبوت
Timescale Cont		
Length Scale O	Conservative	ICLANA
Time and a Fact	10	
Maximum		
	1 mescale	
Convergence C	riteria	
tesidual Type	RMS	•
esidual Target	1.E-4	
Conserva	äon Target	Ŧ
Elapsed Wa	all Clock Time Control	±
	ontroi	E
Interrupt C		
Interrupt C		

Figure 3.10: Solver Settings

3.7.4 Residual Monitors

The residual monitors to be supplied before the solution run are shown in Figure 3.11. At the end of each solver iteration, the residual sum for each of the conserved variables is computed and stored, so recording the convergence history. This information is also included in the data file.



This chapter goes through the project in further depth. This chapter will assist you

in visualising the original concept before deciding on and specifying more specific drawings and sketches. The proposed method for analyzing the design of the Perodua Axia body with several type of spoiler is presented in this chapter. The flow chart and the general process are shown in this chapter as a guided. CATIA software are used to create the design sush as the drawing. The final design in CATIA will be transfer to the Ansys for analyzing and simulation. In addition, the goal of this chapter is to choose the best design to match the project's goals.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

Experimental work is a well-established standard process that have been used to design the Perodua Axia body. This chapter will describe the result for conducting the experimental and simulation. The data from the simulation will be analyze through this chapter. Analysis comparison of the Perodua Axia body several types of spoiler will be show in this chapter. The outcomes and data analysis of the project outcome will be discussed in this chapter. In order to analyze any data, the simulation approach will be used. Ansys software is utilized for the major characteristics of analysis in order to achieve better results. Because it performs structural assessments on fully featured CAD assemblies in minutes, Ansys is a game-changing simulation solution for designers, engineers, and analysts. It eliminates the two most time-consuming, labor-intensive, and error-prone activities in a standard structural simulation: geometry preparation and meshing.

4.2 Results

4.2.1 Velocity Streamlines

In Figure 4.1, it shows the velocity streamline around car body with spoiler A from Ansys software simulation. An 80km/hr. of velocity inlet has been applied. So, this figure 4.1 shows the results of velocity streamlines on the Perodua Axia with spoiler A. It also shows the pattern of aerodynamics external flow happen on the body. On the result, the velocity of streamlines has reach 4.421e+001 which is higher than Perodua Axia with spoiler B.



Figure 4.1: Velocity streamlines around car body with spoiler A

In Figure 4.2, the velocity of streamlines is lower than Perodua Axia with spoiler A which is 4.364+001. The flow pattern of aerodynamics also different on the rear car for both bodies. It shows the recirculation zone that happen on the car. In figure 4.3 the recirculation zone below the spoiler was almost gone. It was a big different as compare in figure 4.1. So, it does show the effect of different type of spoiler will occur different aerodynamics performance.



Figure 4.2: Velocity streamlines around car body with spoiler B

4.2.2 Static Pressure

ALAYS ..

A static pressure comparison is presented in Figure 4.3 and Figure 4.4. It demonstrates how the type of spoiler affects the pressure distribution on the top and front sides of the automobile body, but mostly on the rear side where the spoiler is located. The pressure on Perodua Axia with Spoiler A is lesser compare to Perodua Axia with Spoiler B. The static pressure on Perodua Axia with Spoiler A has reached 4.715e+002 Pa.

A static pressure on Perodua Axia with Spoiler B is 4.736e+002. Even the Perodua Axia with Spoiler B shows the higher pressure, the different of the pressure are not to much different for both bodies



Figure 4.3: Static Pressure on Perodua Axia with Spoiler A



-Figure 4.4: Static Pressure on Perodua Axia with Spoiler B UNIVERSITI TEKNIKAL MALAYSIA MELAKA

4.2.3 Velocity Contour

Contours of velocity for both Perodua Axia with spoiler A and B are shown in Figure 4.5 and Figure 4.6. There were recirculation zones discovered behind the spoiler's rear end. At the back end of both bodies, there were huge and double air swirls. From the both figures, velocity contour on Spoiler B are much faster that Spoiler A. So, the recirculation zone on the rear car does affect the velocity of the car. The velocity contour by Perodua Axia with spoiler has reached 4.491e+001 compare to Perodua Axia with spoiler B which is 4.432e+001.



Figure 4.5: Velocity Contour on Perodua Axia with Spoiler A



UNIVERSITI TEKNIKAL MALAYSIA MELAKA

4.3 Discussion

4.3.1 Drag Coefficient

Model Car	Drag Coefficient
Without Spoiler	0.936
Spoiler A	0.854
Spoiler B	0.853

Table 4.1: Drag Coefficient of car body

Model Car	Drag Force
Without Spoiler	2.82E+02
Spoiler A	2.58E+02
Spoiler B	2.57E+02

 Table 4.2: Drag Force on Car Body

Reducing the drag coefficient of an automobile improves the vehicle's performance in terms of speed and fuel efficiency. A low drag coefficient is connected with high peak speeds and low fuel consumption, but a higher drag coefficient is linked with cars pursuing rapid turning speeds affected by downforce. It explains how the coefficient of drag of an automobile may be computed by evaluating the drag force acting on the vehicle at a certain speed. As a result, the Perodua Axia with Spoiler B has a lower drag coefficient than the Perodua Axia with Spoiler A.



Figure 4.7: Drag Coefficient of Car Body



Figure 4.8: Drag Force on Car Body

4.3.2 Lift Coefficient

Model Car	Lift Coefficient
Without Spoiler	8.60E-02
Spoiler A	2.97E-02
Spoiler B	1.69E-02

Table 4.3: Lift Coefficient of Car Body

Model Car	Lift Force
Without Spoiler	2.60E+01
Spoiler A	8.90E+00
Spoiler B	2.20E+00

Table 4.4: Lift Force on Car Body

The value of lift coefficient is very important because it determines the minimum speed at which car can fly. As the automobile speeds, the air surrounding it moves quicker, producing the vertical force that lifts it off the ground. As the elevating force grows, it reaches a point where the force of gravity is neutralized and the car's tip is raised up. . So, Perodua Axia with Spoiler B does have better lift coefficient which is 1.69E-02 compare to the Perodua Axia without spoiler and Perodua Axia with spoiler A.







Figure 4.10: Lift Force on Car Body

4.4 Chapter Summary

This chapter describes the methodology's findings as well as the conclusions reached as a result of them. Different graphical ways are chosen that visually display the fluid flow structure across the body of the Perodua Axia in order to obtain an airflow field visualization. These techniques include pressure contours, velocity streamlines, and velocity vectors. Aside from qualitative inspection of the fluid flow pattern, statistical method of the simulation results is also performed through monitoring the lift and drag forces operating on the body of the Perodua Axia.



CHAPTER 5

CONCLUSION AND RECOMMENDATION

The project's results and recommendations for additional research were summarized in this chapter. Overall, there has been a lot of study on the computational fluid dynamics of spoilers. An outline of the research as well as a summary of the findings are included in the conclusion section. The section on recommendations contains suggestions for further investigation and improvement of this project.

5.1 Conclusion ALAYSIA

The initial purpose of this study is to use computational fluid dynamics to investigate aerodynamic performance. Essentially, this aim was achieved by using Catia software to create and develop a model of Perodua Axia. The Perodua Axia was designed with two different types of spoilers. Different types of spoilers, according to the findings, have an influence on the car's aerodynamics.

Finally, the finite element study in the Ansys programmed was done successfully to test and compare both types of spoilers. The study of both spoiler such as pressure contours, velocity streamline, and velocity vector in the previous chapter reveals that the overall comparison of the samples demonstrates that the analysis of both spoiler such as pressure contours, velocity streamline, and velocity vector. The spoiler B, according to the analysis, has the chance of becoming the best aerodynamic performance.

5.2 **Recommendations**

With all of the data, this project may constantly expand; here are some suggestions for the future of this project:

- Fabricating light-weight 3D-printed car with spoiler using Selective Laser Sintering (SLS)
- Conducting the real physical test toward the car spoiler instead of doing software analysis.
- Considering using different segment of car for comparison analysis.


REFERENCES

V.K. Naveen, Dr. K.N Lalit, L. N. V.R. Narasimha, Y.R. Sri (2015)

Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler Using CFD. Avaiable at:

https://pdfs.semanticscholar.org/3e35/90f5c1518fd4341cec396a95a4b53ce75a24.pdf

C.Mustafa (2012) CFD study on aerodynamic effects of a rear wing/ spoiler on a passenger vehicle. Avaiable at:

https://scholarcommons.scu.edu/cgi/viewcontent.cgi?referer=https://scholar.google.com/& httpsredir=1&article=1000&context=mech_mstr

K.Joseph (2016) Automotive Aerodynamics. 1st edn. San Diego State University, USA.

T.Chien-Hsiung, F.Lung-Ming, T.Chang-Hsien, H.Yen-Loung, L.Jik-Chang (30 December 2008) Avaiable at: <u>https://www.sciencedirect.com/science/article/pii/S0307904X0800334X</u>

C.D. Rubel, R.Mahmud (19 September 2017) Avaiable at: https://www.sciencedirect.com/science/article/pii/S1877705817332800

B. Bert, T. Yasin (14 July 2015) Avaiable at: https://www.sciencedirect.com/science/article/abs/pii/S0167610515001531

T.Senan, T. H. Thabit (July 2018) Avaiable at: https://portal.arid.my/Publications/ijsrp-p7979190946215.pdf

S. Thomas (2016) Aerodynamics of Road Vehicles. 5th edn.

L.Jim (August 20,2014) LIVE SCIENCE. Avaiable at: https://www.livescience.com/47446-fluid-dynamics.html Sumantran, V, and Sovran G., "Vehicle Aerodynamics," SAE PT-49, Warrendale, PA 1996.

Hucho, H., 1998, Aerodynamics of Road Vehicles, 4th edition, SAE International, Warrendale, PA. 1998.

Katz, J., "Race Car Aerodynamics: Designing for Speed," Bentley Publishers, Second Edition, Cambridge, MA, 2006.

Milliken F., and Milliken M.L., 'Race Car Vehicle Dynamics,' SAE International, Warrendale, PA, 1995.

Katz, J. "Automotive Aerodynamics," Wiley and Sons, Hoboken NJ, April 2016.

Kim, D.-H., & Jo-WonChang. (2014, January). Low-Reynolds-Number Effect On The Aerodynamic Characteristics of a Pitching NACA 0012 Airfoil. Aerospace Science and Technology, 32(1). doi:10.1016/j.ast.2013.08.018

Wang, S., Zhou, Y., Alam, M. M., & Yang, H. X. (2013). Turbulent Intensity Effect on Low Reynolds Number Airfoil Wake. In Fluid-Structure-Sound Interactions and Control (pp. 197-202). Springer Berlin Heidelberg. doi:10.1007/978-3-642-40371-2_29

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Prospathopoulos, J. M., Papadakis, G., Sieros, G., Voutsinas, S. G., Chaviaropoulosm, T. K., & Diakakis, K. (2014). Assessment of the aerodynamic characteristics of thick airfoils in high Reynolds and moderate Ma numbers using CFD modeling. Journal of Physics: Conference SeriesEmail alert RSS feed, 524(1). doi:10.1088/1742-6596/524/1/012015

Muhammad, G. F. (2010). The Effects of Cylinder AT The Leading Edge of Airfoil NACA 0012 On The Boundary Layer Separation. Kufa journal of

J. Bertin, "Aerodynamics for Engineers", Prentice Hall; 5th edition, New Jersey, June 2008

J.Tu, G.H.Yeoh, C.Liu, "Computational Fluid Dynamics: A Practical Approach", Butterworth-Heinemann; 1st edition, Burlington, MA, November 2007 O.Zikanov, "Essential Computational Fluid Dynamics", John Wiley & Sons, Inc. Hoboken, New Jersey, March 2010

K.Gersten, E.Krause, H.Jr.Oertel, C.Mayes "Boundary-Layer Theory", Herrmann Schlichting, 8th Edition, Springer 2004

K.Masaru, N.Tsunehisa, H.Naoki, "Research on Aerodynamic Drag Reduction by Vortex Generator", Mitsubishi Motors Technical Reviews, Tokyo, Japan, 2004

A.Habib, M.I.K.Shah (2019) Applied Fluid Mechanics Lab Manual. 1ts edn.

W.Martin (1996) Fluid Mechanics. 4th edn.

Yang, Z., P. Jia, W. Liu, and H. Yin. 2017. "Car Ownership and Urban Development in Chinese cities: A Panel Data Analysis." Journal of Transport Geography 58: 127-134.

Steg, L. 2005. "Car Use: Lust and Must. Instrumental, Symbolic and Affective Motives for Car Use." Transportation Research Part A: Policy and Practice 39 (2–3): 147–162. السبام ر ويبوم سيتي مالالت

10

Schuitema, G., L. Steg, and S. Forward. 2010. "Explaining Differences in Acceptability before and Acceptance after the Implementation of a Congestion Charge in Stockholm." Transportation Research Part A: Policy and Practice 44 (2): 99–109.

APPENDIX



Appendix A: Orthographic View





UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA

TAJUK: COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF PERODUA AXIA BODY WITH SEVERAL TYPES OF SPOILER

SESI PENGAJIAN: 2020/21 Semester 1

Saya WAN MUHAMMAD 'AFIF BIN WAN AZIR

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 (

SULIT

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

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

TIDAK TERHAD

Disahkan oleh:

18/1/2022

WAN MUHAMMAD AFIF BIN WAN AZIR Alamat Tetap:

Cop Rasmi:

Dr Setyamartana Parman Senior Lecturer Faculty of Machanical & Manufacturing Engineering Technolo Universiti Teknikal Malavsia Melaka

287, TAMAN PERTAMA, 27200

KUALA LIPIS, PAHANG

Tarikh: 17/1/2022

Tarikh:

** 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.



 Universiti Teknikal Malaysia Melaka Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia. (2) +606 270 1000
(2) +606 270 1022

@ www.utem.edu.my

FAKULTITEKNOLOGI KEJURUTERAAN MEKANIKAL DAN PEMBUATAN

Tel : +606 270 1184 | Faks : +606 270 1064

Rujukan Kami (Our Ref): Rujukan Tuan (Your Ref): Tarikh (Date): 31 Januari 2021

Chief Information Officer Perpustakaan Laman Hikmah Universiti Teknikal Malaysia Melaka

Melalui

Dekan Fakulti Teknologi Kejuruteraan Mekanikal dan Pembuatan Universiti Teknikal Malaysia Melaka

Tuan

WALAYSIA

PENGKELASAN TESIS SEBAGAI TERHAD BAGI TESIS PROJEK SARJANA MUDA

Dengan segala hormatnya merujuk kepada perkara di atas.

 Dengan ini, dimaklumkan permohonan pengkelasan tesis yang dilampirkan sebagai TERHAD untuk tempoh LIMA tahun dari tarikh surat ini. Butiran lanjut laporan PSM tersebut adalah seperti berikut:

Nama pelajar: WAN MUHAMMAD 'AFIF BIN WAN AZIR (B091810444) Tajuk Tesis: COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF PERODUA AXIA BODY WITH SEVERAL TYPES OF SPOILER

3. Hal ini adalah kerana IANYA MERUPAKAN PROJEK YANG DITAJA OLEH SYARIKAT LUAR DAN HASIL KAJIANNYA ADALAH SULIT SIA MELAKA

Sekian, terima kasih.

"BERKHIDMAT UNTUK NEGARA" "KOMPETENSI TERAS KEGEMILANGAN"

Saya yang menjalankan amanah,

Amtomy Dr Setyamartana Parman Senior Lecturer Feedby Menanica & Manisterium Grigowang Technolog University Einikal Malaysia Melaka

Penyelia Utama/ Pensyarah Kanan Fakulti Teknologi Kejuruteraan Mekanikal dan Pembuatan Universiti Teknikal Malaysia Melaka





Submission date: 17-Jan-2022 01:40PM (UTC+0800) Submission ID: 1742834225 File name: B091810444_FullThesis.pdf (1.67M) Word count: 11518 Character count: 58121

COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF PERODUA AXIA BODY WITH SEVERAL TYPES OF SPOILER



mechanixillustrated.technicacuriosa.com

8

Senarai semak Tesis Muktahir PSM (Library Version)

Nama Pelajar	:	
Program	:	No. Matrik:
Nama Penyelia	:	

	SENARAI SEMAK	Yes	No
1.	FRONT PAGE (All Caps, Arial Narrow, Bold, 18, Centre Alignment - margin (top 4		
	cm, bottom 2.5 cm, left 3 cm and right 2.5 cm)		
	UTeM Logo:		
	TITLE: maximum 4 line		
	STUDENT NAMES: maximum 2 line		
	STUDENT MATRIC NO. : no spacing with Student Name		
	PROGRAM: BACHELOR OF XXX ENGINEERING TECHNOLOGY (program here)		
	WITH HONOURS		
	YEAR: 2022		
2.	FACULTY FRONT PAGE (Times New Roman, Bold, Centre alignment) - margin		
	(top 4 cm, bottom 2.5 cm, left 3 cm and right 2.5 cm)		
	UTeM Logo:		
	Faculty: Faculty of Mechanical and Manufacturing Engineering Technology		
	(Capitalized Each Word, 14)		
	Title: All caps, maximum 4 line Times New Roman, 14		
	Student Names: Capitalized Each Word, maximum 2 line Times New Roman, 12		
	Program : Bachelor Of XXX Engineering Technology (Write Your Program Here) With		
	Honours (Capitalized Each Word, Times New Roman, 12)		
	Year: 2022 (the submission year),12		
3.	PSMFRONT PAGE (Times New Roman, Bold, Centre alignment - margin (top 4 cm,		
	bottom 2.5 cm, left 3 cm and right 2.5 cm)		
	Title: ALL CAPS, maximum 4 line, 14		
	Student Names: ALL CAPS, maximum 2 line, 12		
	Thesis of Program: A thesis submitted in fulfillment of the requirements for the degree		
	of Bachelor of XXX Engineering Technology (Write your program here) with		
	Honours (Times New Roman, 12) EKNIKAL MALAYSIA MELAKA		
	Faculty: Faculty of Mechanical and Manufacturing Engineering Technology		
	(Capitalized Each Word, Times New Roman, 12)		
	University: UNIVERSITI TEKNIKAL MALAYSIA MELAKA (ALL CAPS, Times		
	New Roman, 12)		
	YEAR: 2022 (Times New Roman, 12)		
4	DECLARATION Justify margin (top 4 cm, bottom 2.5 cm, left 3 cm and right 2.5		
	cm), Capitalize each word, Times New Roman 12,		
_	digital Signature		
5	APPROVAL Justify margin (top 4 cm, bottom 2.5 cm, left 3 cm and right 2.5 cm),		
	Capitalize each word, Times new roman. 12,		
_	digital Signature		
6	DEDICATION Justify margin (top 4 cm, bottom 2.5 cm, left 3 cm and right 2.5 cm),		
-	These New Koman, 12, Double spaced		
/	ABSTRACT (English version) Justity margin (top 4 cm, bottom 2.5 cm, left 3 cm and		
	right 2.5 cm), Not more than I page and no multiple paragraphs, Times New Roman,		
0	12, Single spaced		
δ.	ADDIKAN (<i>Malay version</i>) Justify margin (top 4 cm, bottom 2.5 cm, left 3 cm and night 2.5 cm). Not more than 1 page and no multiple a supervised by Times N		
	right 2.5 cm), Not more than 1 page and no multiple paragraphs, 1 mes New Roman,		
	Halle, 12, Single spacea		
9	ACKING W LEDGENIEN IS Justify margin (top 4 cm, bottom 2.5 cm, left 3 cm and		
	right 2.5 cm) limes New Koman, 12, Single or 1.5 spaced		

10	CONTENT Justify margin (top 2.5 cm, bottom 2.5 cm, left 3 cm and right 2.5 cm)	
	Line and Paragraph 2.0 spaced	
	Table number (chapter. No of table) before Table Title, just separate the title number	
	and and the title using ONLY ONE tab	
	Figure number (chapter. No of figure) before Figure Title, just separate the title	
	number and and the title using ONLY ONE tab	
11.	BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA	
	• Student signature,	
	SV digital signature & Stamp	
12.	Turnitin Report – dilampirkan,	
	SV digital signature & Stamp	

Ulasan:

Disahkan: Semakan: AJK Tesis Muktahir PSM UNIVERSI Pengerusi JKPPU (PSM) AYSIA MELAKA MAL

Catar alir Semakan Tesis Muktahir PSM (Library Version)

