Bincang Sains dan Teknologi (BST) **E-ISSN 2961-8746**
DOI: 10.56741/bst.v5i01.2026 **P-ISSN 2961-8932**



The Effect of the Number of Fins on Rocket Aerodynamic Performance Using Computational Fluid Dynamics

[1]Mohamad Yamin*, [1]Nuril Hadi
Corresponding Author: * mohay@staff.gunadarma.ac.id
[1] Gunadarma University, Jakarta, Indonesia

Article submitted: 26 January 2026 **Revised:** 28 February 2026 **Accepted:** 18 March 2026

Abstract

This study aims to analyze the effect of the number of fins on the aerodynamic characteristics of a rocket using the Computational Fluid Dynamics (CFD) method. Three rocket configurations with different numbers of fins (3, 4, and 6 straight fins) were modeled and simulated at a subsonic speed of Mach 0.6 with angles of attack ranging from 0° to 25°. The main aerodynamic parameters analyzed include the lift coefficient (Cl), drag coefficient (Cd), and moment coefficient (Cm). The simulation results show that increasing the number of fins tends to improve rocket stability and produces a more uniform pressure distribution. The six-fin configuration yields the highest Cl/Cd ratio, particularly at an angle of attack of 10°, indicating the best aerodynamic efficiency. In addition to aerodynamic performance, economic considerations are also an important factor; the use of six fins provides advantages in terms of stability, although it requires higher manufacturing costs compared to three or four fins. Overall, the six-fin configuration is recommended for rocket applications that req uire high stability and aerodynamic efficiency.

Keywords: *Aerodynamics, ANSYS, CFD, Fins, Rocket.*

Introduction

Rockets are one of the major technological achievements that have advanced rapidly throughout the history of engineering and are widely used in various fields such as military applications, space exploration, meteorology, and satellite launches. Rocket propulsion systems rely on thrust generated by fuel combustion; however, aerodynamic performance remains a crucial factor in flight efficiency and stability, particularly during atmospheric flight [1]. The purpose of this study is to compare the aerodynamic performance of rockets equipped with 3, 4, and 6 fins under subsonic conditions. By conducting CFD simulations using ANSYS Fluent, this research aims to determine the optimal fin-number configuration in terms of aerodynamic efficiency and flight stability.

External components such as the nose cone, rocket body, and fins play an important role in generating aerodynamic forces such as lift, drag, and moment. Among these components, fins function to provide a restoring moment against rotational disturbances, allowing the rocket to return to a stable trajectory [2]. Fin shapes and configurations vary depending on the mission and design requirements, ranging from straight to curved geometries [3],[4].

Several studies have compared the aerodynamic performance of straight and curved fins. Barokah et al. [3] showed that straight fins provide more stable lift and drag coefficients over variations

in angle of attack. However, the effect of the number of fins on stability and aerodynamic efficiency has not been comprehensively investigated. In practice, rockets with three fins are lighter and have lower drag, but they may experience lateral imbalance. In contrast, four fins provide a more symmetric force distribution, while six fins enhance stability but increase drag and structural load [5].

Advances in numerical simulation methods such as CFD enable the evaluation of aerodynamic performance without direct testing in wind tunnels [6]. Software packages such as ANSYS Fluent are capable of calculating pressure distributions, velocity vectors, and aerodynamic coefficients with a high level of precision [7],[8]. CFD is also widely used to evaluate moments and lift forces under variations in angle of attack and fin configurations [9].

Although several studies have discussed variations in fin shape [3], asymmetric positioning [10], and the use of micro-scale elements such as microflaps [4], direct comparative studies on the number of fins under identical geometric and velocity conditions remain limited.

Therefore, this study aims to numerically investigate the effect of the number of fins (3, 4, and 6) on the aerodynamic characteristics of a rocket using CFD simulations. The study is conducted at a subsonic–transitional speed (Mach 0.8) with angles of attack ranging from 0° to 20°, while maintaining uniform fin and rocket body geometries. The results are expected to provide quantitative insight into the influence of fin configuration on force efficiency and directional stability, and to serve as a design reference for small-scale rockets and other launch vehicles.

In practice, the fin-number configuration plays a crucial role in determining the aerodynamic characteristics of a rocket. A three-fin configuration is often used to achieve mass efficiency and minimal drag; however, it may introduce lateral imbalance. A four-fin configuration is a common choice because it provides a symmetric force distribution and improved stability. Meanwhile, a six-fin configuration, although offering high directional stability, can increase structural loads and raise the drag coefficient [2],[11]. Unfortunately, systematic comparisons of the aerodynamic performance among three-, four-, and six-fin configurations under identical conditions and geometries are still scarce in the scientific literature.

On the other hand, advances in computational technology have enabled aerodynamic analysis to be conducted without full reliance on wind tunnel testing or direct launch experiments through the use of CFD [12]. This method employs numerical approaches to solve the Navier–Stokes equations for fluid flow, allowing high-accuracy and cost-efficient visualization of forces and flow fields around objects [13]. The use of software such as ANSYS Fluent in rocket simulations has become a standard practice in the research and development of modern flight vehicles, including the evaluation of parameters such as drag coefficient (Cd), lift coefficient (Cl), and moment coefficient (Cm) under various flight conditions.

This study aims to fill the existing research gap by investigating the effect of the number of fins (3, 4, and 6) on the aerodynamic characteristics of a rocket using CFD simulations. By maintaining constant fin geometry and rocket dimensions, and varying the angle of attack from 0° to 20° at a subsonic–transitional speed (Mach 0.8), this research seeks to provide a comprehensive understanding of the advantages and limitations of each configuration. The primary focus of this study is to compare the drag coefficient (Cd), lift coefficient (Cl), moment coefficient (Cm), and the Cl/Cd ratio as indicators

of aerodynamic efficiency and stability. The results of this study are expected to serve as a reference for rocket designers and small-scale launch vehicle developers in selecting the optimal fin configuration in terms of fuel efficiency, directional stability, and maneuverability within the atmosphere.

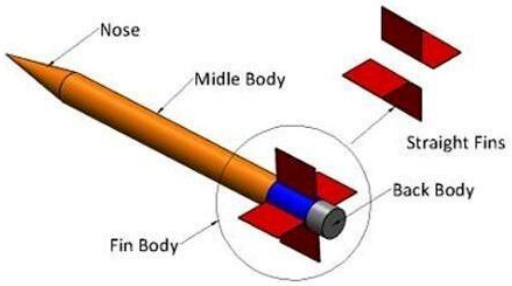


Fig. 1. Components of a rocket

Rockets, including spacecraft, missiles, and other aerial vehicles, obtain thrust from the reaction produced by the combustion of propellants. This thrust is generated through rapid combustion or explosive reactions within the rocket combustion chamber, where one or more types of fuel are combined. The propellant, which acts as the driving agent, undergoes combustion in a dedicated chamber, producing extremely high pressure and temperature. This thermal energy and pressure are then directed through a nozzle located at the rear of the rocket, generating thrust that propels the rocket forward in accordance with Newton's action–reaction principle [1],[14]. As shown in Fig. 2, a rocket generally consists of four main systems: the structural system, payload system, guidance system, and propulsion system. These four systems play essential roles in maintaining structural integrity, ensuring accurate flight direction, and guaranteeing mission success [11].

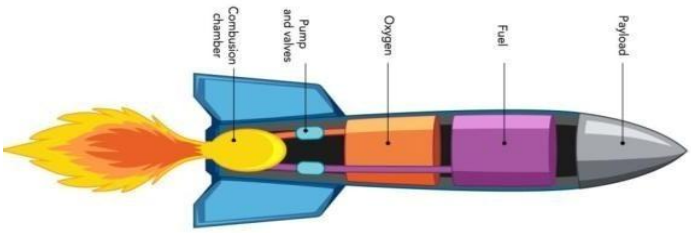


Fig. 2. Rocket components

During its flight trajectory, a rocket experiences changes in attitude, a phenomenon caused by the interaction of internal and external forces and moments, as illustrated in Fig. 3. The internal forces and moments generally originate from the propulsion system, which may produce asymmetric thrust or small fluctuations due to non-uniform combustion. On the other hand, external factors causing changes in rocket attitude arise from gravitational forces as well as aerodynamic moments generated as the rocket travels through the atmosphere [1].

These aerodynamic moments arise from the interaction between the rocket's geometric shape and the surrounding airflow. As the rocket travels through the atmosphere, it is subjected to various forces such as dynamic pressure, frictional forces, lift, and drag. The magnitude of these forces strongly depends on the rocket's geometric configuration and surface finishing techniques, which can influence whether the flow around the rocket is turbulent or laminar [2].

E-ISSN 2961-8746
P-ISSN 2961-8932

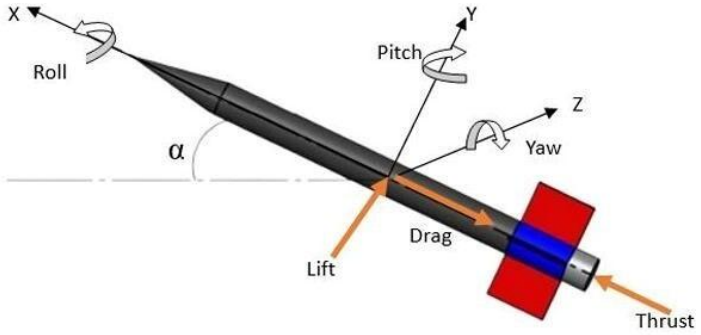


Fig. 3. Rocket degrees of freedom (DOF)

Numerical aerodynamic studies of rockets using the CFD approach have become a leading method for analyzing aerodynamic force characteristics such as drag, lift, and moments. Through simulations under various flow conditions, including subsonic and supersonic speeds and different angles of attack, researchers can evaluate in detail the influence of fin geometry on the airflow around the rocket [11],[15]. Simulation results indicate that as the angle of attack and flow velocity increase, the drag coefficient (Cd) and lift coefficient (Cl) tend to increase, while the moment coefficient (Cm) tends to decrease [16]. This phenomenon illustrates that changes in flight conditions significantly affect rocket performance and stability, depending on the shape and number of fins used.

In this context, the drag coefficient (Cd) is a dimensionless parameter that represents the magnitude of aerodynamic drag experienced by an object as it moves through a fluid. This drag arises from the interaction between the airflow and the object's surface, involving both pressure drag (form drag) and frictional forces (skin friction) [2]. Meanwhile, the lift coefficient (Cl) indicates the ability of a surface in this case, rocket fins to generate lift. Lift is produced by the pressure difference between the upper and lower surfaces of the fins, which is induced by differences in airflow velocity, as explained by Bernoulli's principle. The upper surface, where the flow velocity is higher, experiences lower pressure than the lower surface, resulting in an upward force that contributes to the rocket's vertical stability [1],[17].

Furthermore, a thorough understanding of these parameters is essential in the early stages of rocket design, particularly for selecting fin configurations that provide maximum efficiency with minimal drag. CFD-based simulations enable accurate visualization and calculation of these aerodynamic parameters without the need for costly physical testing in wind tunnels [10].

$$C_D = \frac{F_D}{\frac{1}{2}\rho v^2 A} \quad (1)$$

$$C_L = \frac{F_L}{\frac{1}{2}\rho v^2 A} \quad (2)$$

In aerodynamics, the term *moment* refers to the rotational motion caused by forces acting on an aircraft or rocket body about its center of gravity (CoG). This rotational effect arises because aerodynamic forces generate torque, which in turn induces angular velocity about the principal coordinate axes of the system [2]. In general, rotational motion in flight dynamics occurs about three axes: the longitudinal axis (X), the lateral axis (Y), and the vertical axis (Z). Each of these rotations produces an angular velocity denoted as P (rolling), Q (pitching), and R (yawing). These angular

velocities are associated with the corresponding aerodynamic moments: L for roll moment, M for pitch moment, and N for yaw moment. These three types of motion—rolling (rotation about the X-axis), pitching (about the Y-axis), and yawing (about the Z-axis)—are integral components of the motion dynamics of a flight vehicle, including rockets.

An understanding of aerodynamic moments is crucial in the rocket design process because these moments directly influence flight stability and directional control [18],[19]. Fin geometry, the position of the center of mass, and the angle of attack strongly determine the magnitude of the moments generated while the rocket travels through the atmosphere.

$$C_M = \frac{F_M}{\frac{1}{2}\rho v^2 A} \quad (3)$$

The numerical simulations employed in this study were carried out using the Finite Volume Method (FVM), combined with a turbulence model to simulate fluid flow under two velocity conditions: Mach 0.6 (subsonic) and Mach 1.2 (supersonic) [20]. The analyzed fluid flow was assumed to be steady, viscous, and isothermal. Under this isothermal condition, no significant temperature variation is assumed to occur throughout the flow domain; therefore, the energy conservation equation was not included in the formulation of the governing equations [13].

Instead, the simulations solved the Reynolds-averaged Navier–Stokes (RANS) equations, in which the effects of turbulence are represented through a two-equation eddy-viscosity closure. Specifically, the turbulence closure solves transport equations for the turbulent kinetic energy, k, and a second scale-determining variable—either the specific dissipation rate, ω, or the turbulent dissipation rate, ε. This class of models is widely used to improve robustness and accuracy in flows exhibiting separation and strong adverse pressure gradients, which are typical in the vicinity of rocket fins [15]. Accordingly, the governing system comprises the conservation of mass and momentum, supplemented by two additional transport equations for the turbulence variables. Steady, incompressible governing equations:

Continuity (mass conservation)

$$\nabla \cdot V = 0, \qquad V = V(x, y, z) \quad (4)$$

Momentum (RANS)

$$\rho(\mathrm{V} \cdot \nabla)V = \nabla p + \nabla \cdot [(\mu + \mu_t)(\nabla V + (\nabla V)^T)] \quad (5)$$

where ρ is density, μ is the molecular dynamic viscosity, and μt is the turbulent (eddy) viscosity provided by the turbulence model.

In component form:

$$\rho V_j \frac{\partial V_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_i}{\partial x_j}\right)\right] \quad (6)$$

Turbulence Kinetic Energy, k

$$\frac{\partial k}{\partial t} + V_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k\omega + \frac{\partial}{\partial x_j}\left[(v + v_t)\frac{\partial k}{\partial x_j}\right] \quad (7)$$

Specific Dissipation Rate, ω

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha \frac{\omega}{k} P_k - \beta\, \omega^2 + \frac{\partial}{\partial x_j}\left[(v + \sigma_\omega v_t)\frac{\partial \omega}{\partial x_j}\right] + 2(1 - F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \quad (8)$$

E-ISSN 2961-8746
DOI: 10.56741/bst.v5i01.2026
P-ISSN 2961-8932

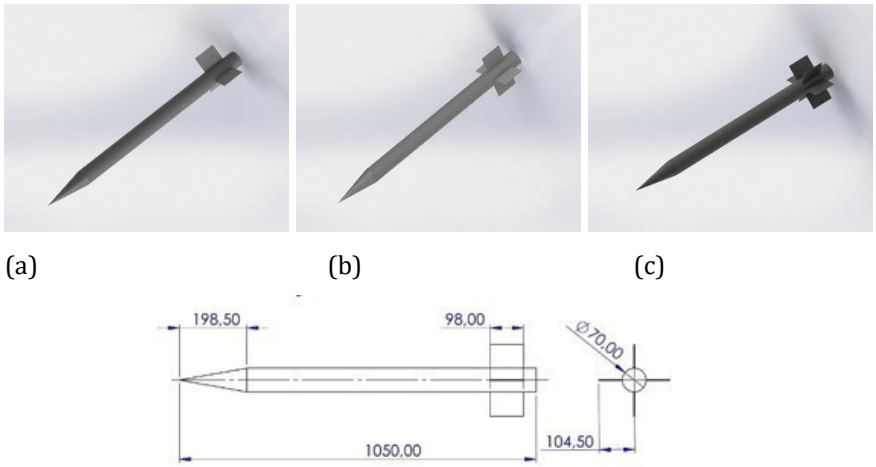
where:

$\nu \equiv \mu/\rho$the kinematic viscosity and $\nu_t \equiv \mu_t/\rho$,
$P_k$the production of turbulent kinetic energy (commonly $P_k = \tau_{ij}\ \partial U_i/\ \partial x_j$, with $\tau_{ij}$the modeled Reynolds stress term),
$\alpha, \beta, \beta^*, \sigma_k, \sigma_\omega, \sigma_{\omega 2}$model constants,
F_1a blending function (as used in Menter-type SST formulations).

Methods

A. Geometry Modeling

The initial step prior to conducting CFD simulations to obtain the drag coefficient (Cd), lift coefficient (Cl), and moment coefficient (Cm) is to construct three-dimensional (3D) models of the rocket with straight fins and the rocket with curved fins. The geometric differences between these two fin types are illustrated in Fig. 4, which clearly presents the visual distinction between the curved-fin and straight-fin configurations.



(a) (b) (c)

(d)

Fig. 4. (a) Three-fin rocket model, (b) four-fin rocket model, (c) six-fin rocket model, and (d) rocket dimensional specifications

To ensure a fair comparison of the simulation results, both rocket models were designed with the same total surface area of 0.269 m^2. This approach was intended to isolate the effect of fin shape on aerodynamic characteristics without bias from surface area differences. The general specifications of the rocket fins, including their dimensions and main components, are presented in detail in Tables 1 and 2. This information includes the chord length, fin height, thickness, and the relative position of the fins with respect to the main rocket body.

Table 1. Rocket Dimensions

Rocket Model (Body)	Specifications
Overall length, **L**	1050 mm
Body diameter, **D**	70 mm
Nose type & length, $\mathbf{L_n}$	Conical, 198.5 mm
Rear body length, $\mathbf{L_a}$	851.5 mm
L/D Ratio	15
Number of fins	3, 4, 6

E-ISSN 2961-8746
P-ISSN 2961-8932

Table 2. Rocket Fin Dimensions

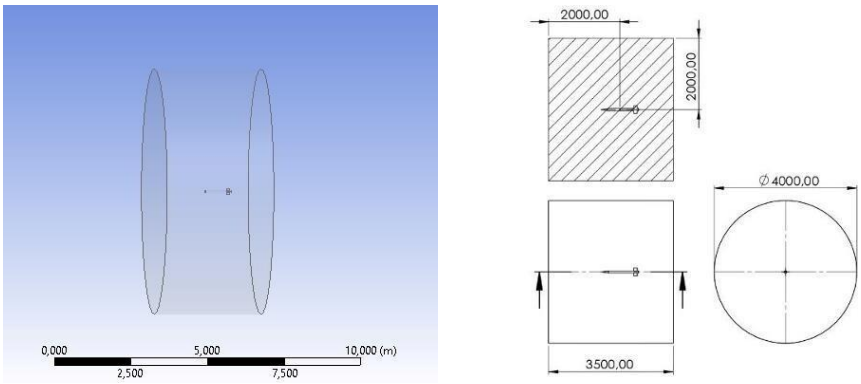
Fin Model	Specifications
Fin Planform	Rectangular
Fin Configuration	Sirip Lurus
Single fin span	69.5 mm
Root chord	98 mm
Fin thickness	2 mm
Fin leading-edge angle	0°

B. Simulation

1. Enclosure

The 3D rocket model that had been created was then imported in STP (.stp) format into the simulation software to generate the numerical fluid domain (enclosure). The size and shape of the enclosure were scaled proportionally to the dimensions of the test object (rocket) to ensure that boundary effects on the simulation results were minimized.

In this study, the fluid domain was designed in a cylindrical shape, as shown in Fig. 5. The domain has a diameter of 4 m and a length of 3.5 m, providing sufficient space for the fluid flow to develop naturally around the rocket without interference from the domain boundaries. These dimensions were selected based on standard practices in CFD simulations to ensure that the interaction effects between the domain walls and the main flow remain negligible



(a) (b)

Fig. 5. (a) Enclosure in ANSYS, (b) enclosure dimensions.

2. Meshing Process

The meshing process (also known as discretization) in CFD simulations is the stage of creating a discrete model of the geometric domain to be analyzed. In this stage, the fluid domain is divided into small elements or cells, referred to as a mesh or grid, enabling numerical calculations based on the fundamental mathematical equations of CFD, such as the conservation of mass and momentum [10],[21].

The use of a large number of mesh elements can improve simulation accuracy by reducing discretization errors. However, excessively fine or dense meshes can significantly increase computational time and data storage requirements. Therefore, a compromise between accuracy and computational efficiency is required, which can be achieved by determining an optimal and stable mesh size through a mesh independence test.

In this study, a mesh independence test was conducted to ensure that the simulation results no longer change significantly as the number of mesh elements is increased. This process is illustrated in Fig. 6, which shows the variation of aerodynamic parameters (such as Cd) with respect to the number of mesh elements. The results of this test were used as a reference to select the final mesh size that provides the best balance between accuracy and computational time.

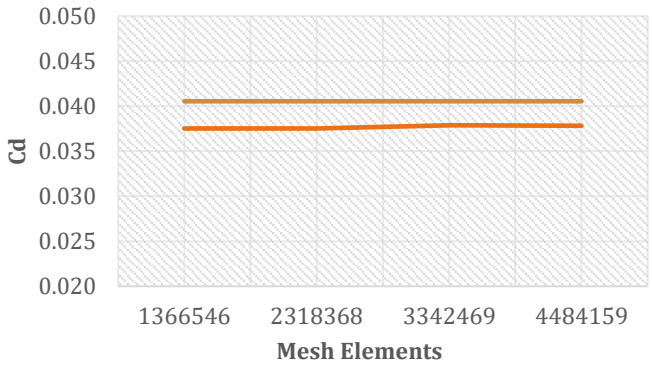


Fig. 6. Rocket Mesh Grid

Table 3. Rocket Mesh Grid Data

mesh	cd (Dahlan, 2017)	cd sim
1366546	0.040540	0.0375
2318368	0.040540	0.0375
3342469	0.040540	0.0379
4484159	0.040540	0.0378

The mesh independence test results showed that mesh sizes of 4,484,159 and 1,366,546 elements produced relatively stable drag coefficient (Cd) values, ranging from 0.0375 to 0.0378. No significant changes in Cd were observed even when the number of mesh elements was reduced. Based on these results, it can be concluded that the simulation has reached mesh-independent conditions. Therefore, a mesh size of 3,342,469 elements was selected as the optimal mesh for the main simulations, as it provides accurate results with good computational efficiency.

The mesh topology used in this study consists of tetrahedral elements, as shown in Fig. 7. This element type was selected due to its ease of meshing and its capability to handle complex geometries, such as curved rocket fins. After mesh treatment and refinement, the mesh quality showed an average skewness value of 0.2282.

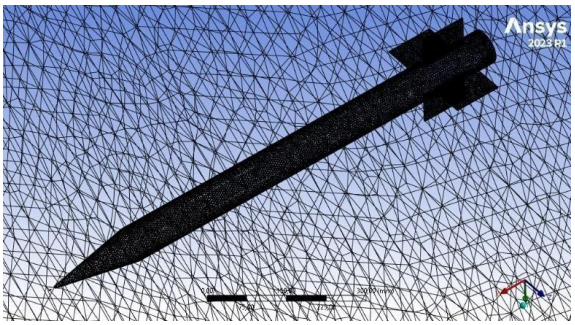


Fig. 7. Tetrahedral Mesh on the Rocket

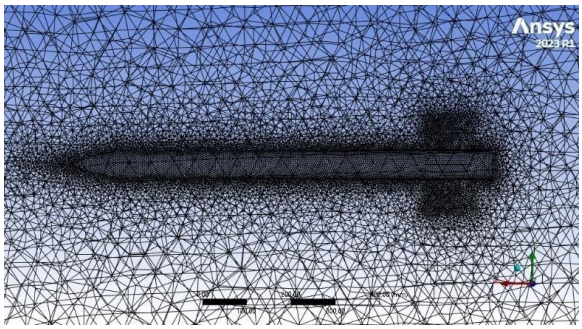


Fig. 8. Tetrahedral Mesh Slice on the Rocket

Skewness values approaching 0 and orthogonal quality values approaching 1 indicate that the mesh quality obtained in this validation process is sufficiently good and numerically acceptable for CFD analysis. High-quality mesh is crucial to ensure that the numerical solution is not affected by element geometry imbalances or grid quality degradation, which could potentially lead to numerical instability.

3. Boundary condition

The boundary conditions in this study followed the approach used by Dahalan et al. (2017), as shown in Fig. 8. The airflow was assumed to be steady and isothermal, with a fluid density of 1.17 kg/m^3 and a dynamic viscosity of 1.7894 × 10^{-5} kg/m·s. At the inlet, a velocity-inlet boundary condition was applied with two flow speeds: Mach 0.6 for subsonic flow and Mach 1.2 for supersonic flow. At the outlet, a pressure-outlet boundary condition was used with a zero-gauge pressure.

To represent turbulent conditions during low-speed flight, the turbulence intensity at the inlet was set to 5%, which has been shown to provide high sensitivity to stall or flow separation phenomena in various numerical and experimental studies [12]. On the outer surfaces of the fluid domain (enclosure), an ideal wall boundary condition with zero shear stress was applied. This approach is used to avoid numerical errors due to the geometric limitations of the fluid domain, even though, physically, it represents an unbounded space [22],[23].

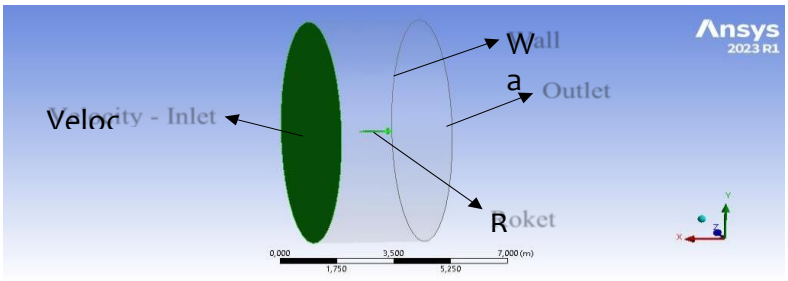


Fig 9. Boundary Conditions

Table 4. Reference Values

Parameter	Value	Unit
Density	1.17	kg/m^3
Cp (Specific Heat)	1006.43	J/(kg·K)
Thermal Conductivity	0.0242	W/(m·K)
Viscosity	1.7894 × 10^{-5}	kg/(m·s)
Molecular Weight	28.966	kg/kmol
Velocity Inlet	204.174 and 408.348	m/s
Turbulent Intensity	5	%
Turbulent Viscosity Ratio	10	–
Pressure Outlet	0	Pa
Temperature	288,16	K
Length	1	m
Area	0,049875	m^2
Rocket Wall	No-slip condition	–

In this simulation, the rocket surface was defined as a stationary wall with a no-slip condition. The application of the no-slip condition results in the fluid velocity at the rocket surface being zero relative to the solid surface, in accordance with the log-law of the wall, which describes the velocity distribution caused by shear stress in the boundary layer on the rocket surface [13]. This

phenomenon is crucial for analyzing flow formation and drag accumulation due to fluid surface interactions.

To solve the numerical problem of turbulent flow, this study employed the k–omega SST (Shear Stress Transport) turbulence model with a coupled solution scheme. The k–ω SST model was selected for its ability to predict flow separation along the rocket surface and its proven reliability through validation against experimental data [12]. Additionally, this model is well-suited for handling near-wall turbulent flows, making it appropriate for aerodynamic simulations of complex geometries such as rockets with curved or straight fins.

Results and Discussion

A. Pressure Distribution

The CFD simulation results for rockets with 3, 4, and 6 fins at Mach 0.6 and angles of attack ranging from 0° to 25° are shown in Fig. 9. From the graphs, it can be observed that the drag coefficient (Cd) and lift coefficient (Cl) increase with increasing angle of attack, while the moment coefficient (Cm) tends to decrease. Among the three configurations, the three-fin rocket exhibits the lowest Cd, followed by the four-fin rocket, with the six-fin rocket having the highest drag. Conversely, the highest Cl is found in the six-fin rocket and the lowest in the three-fin rocket. The Cm of the six-fin rocket also shows a more negative value, indicating better stability. Overall, the four-fin rocket demonstrates a balanced combination of aerodynamic efficiency and stability.

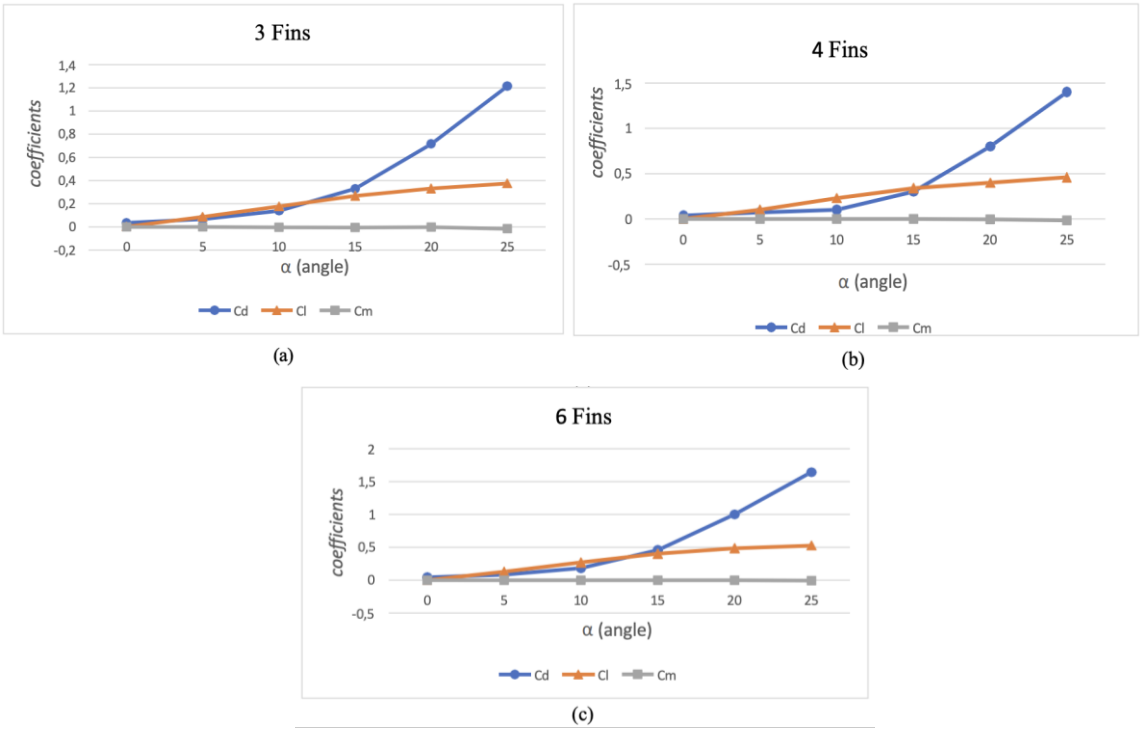


Fig. 10. (a) Mach 0.6, three-fin rocket, (b) Mach 0.6, four-fin rocket, (c) Mach 0.6, six-fin rocket

The CFD simulation results for rockets with 3-, 4-, and 6-fin configurations at Mach 0.6 and angles of attack (AoA) ranging from 0° to 25° show a significant influence on aerodynamic coefficients. As presented in Fig. 13, the drag coefficient (Cd) generally increases with increasing AoA. For the four-fin

configuration, Cd rises from 0.037 at 0° AoA to 1.4 at 25° AoA, indicating that the four-fin setup produces the highest drag among the configurations. The three-fin configuration shows an increase in Cd from 0.0355 to 1.2444, while the six-fin configuration increases from 0.0423 to 1.2453. These results indicate that the three-fin configuration has the lowest initial Cd and tends to maintain aerodynamic efficiency at low to moderate angles of attack, although at higher AoA the differences in Cd among the configurations become smaller.

The lift coefficient (Cl) also shows an increase in line with the rising angle of attack. For the four-fin configuration, Cl increases from 0 at 0° AoA to 0.46 at 25° AoA. The three-fin configuration rises from 0 to 0.7346, while the six-fin configuration increases from 0 to 0.7432. Thus, it can be concluded that the six-fin configuration provides the highest lift at the maximum angle of attack, although the difference from the three-fin configuration is not very large. The four-fin configuration produces the lowest lift, indicating that despite its higher drag, the increase in lift is not proportionate.

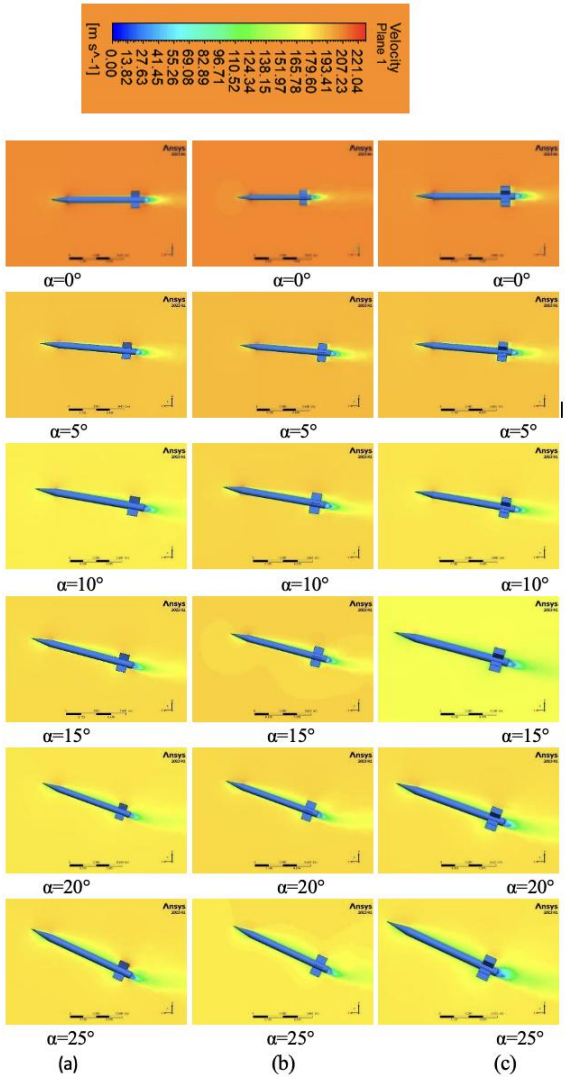


Fig. 11. (a) Mach 0.6 velocity contour for the three-fin rocket, (b) Mach 0.6 velocity contour for the four-fin rocket, (c) Mach 0.6 velocity contour for the six-fin rocket

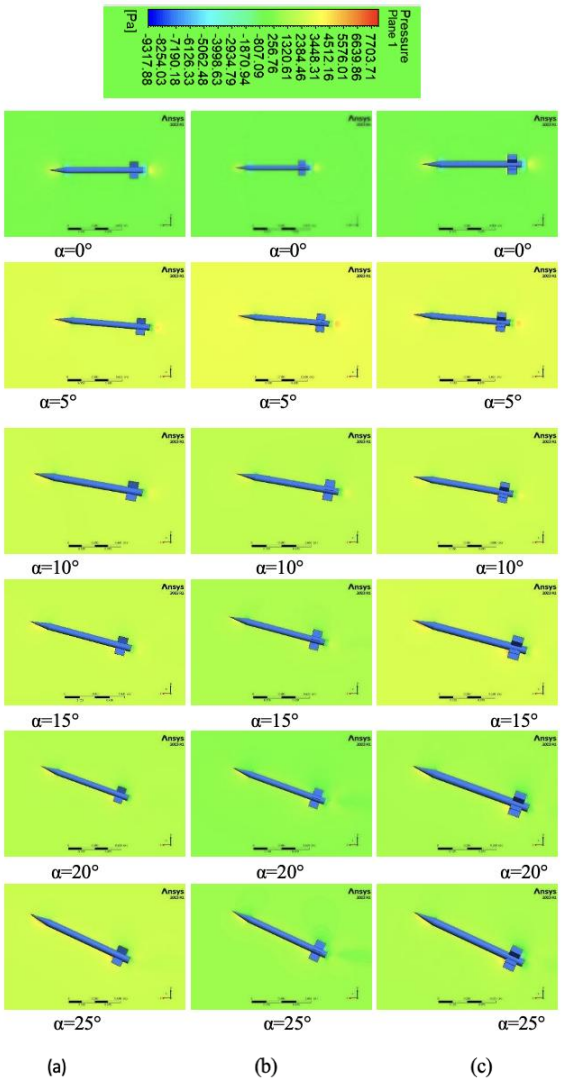


Fig. 12. (a) Mach 0.6 pressure contour for the three-fin rocket, (b) Mach 0.6 pressure contour for the four-fin rocket, (c) Mach 0.6 pressure contour for the six-fin rocket

E-ISSN 2961-8746
P-ISSN 2961-8932

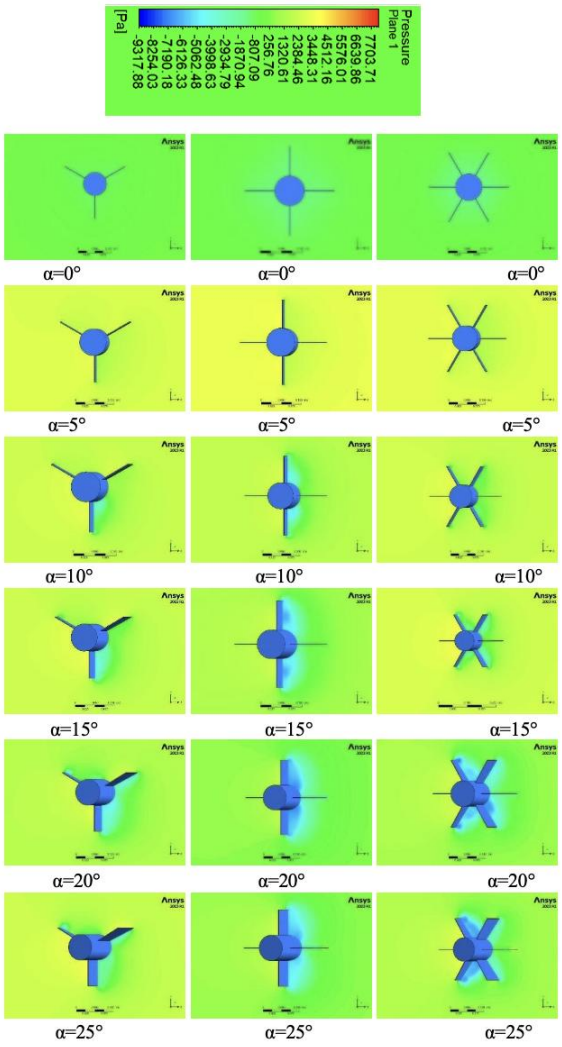


Fig 13. Pressure Contour on the Fin Section

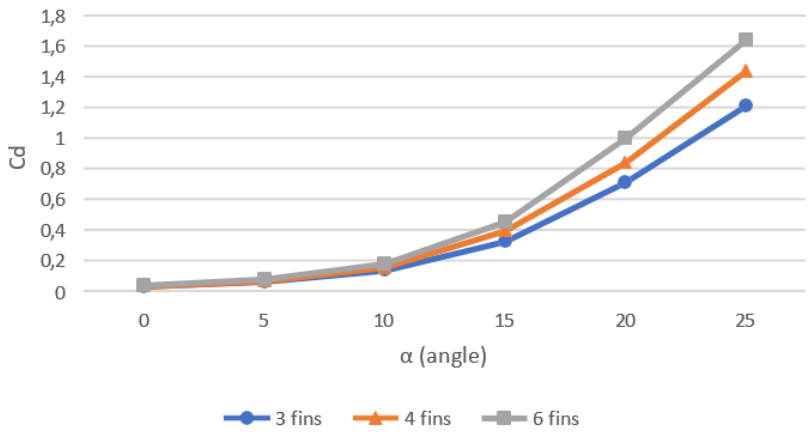


Fig. 14. Cd values for 3,4, and 6 fins

Meanwhile, the moment coefficient (Cm) decreases significantly for all configurations, indicating a tendency for nose-down pitching at high angles of attack. For the four-fin configuration, Cm drops from -0.0004 to -0.016. The three-fin configuration experiences a more extreme decrease from -0.0018 to -0.9008, while the six-fin configuration decreases from -0.0001 to -0.9044. This means that although the three and six-fin configurations generate high lift, directional stability (moment control) becomes more challenging at large angles of attack.

To validate the simulation results in this study, a comparison was made with the research conducted by Dahalan et al. [11], which investigated the aerodynamic characteristics of rockets with curved fins using a semi-empirical approach and numerical simulations. Their study compared results from wind tunnel tests, the USAF DATCOM software, and numerical simulations to assess the accuracy of the rocket aerodynamic models.

Unlike that study, the present research focuses solely on straight fins and compares the effects of different fin numbers 3, 4, and 6 fins. Simulations were conducted using CFD to evaluate the aerodynamic performance of each configuration. Validation was performed by comparing the drag coefficient (Cd) obtained from the simulations with reference data from Dahalan et al. [11]. Fig. 14 shows the comparison of Cd values between this study and the previous simulations, indicating that the errors fall within acceptable limits. Thieinforces the validity of the simulation approach used.

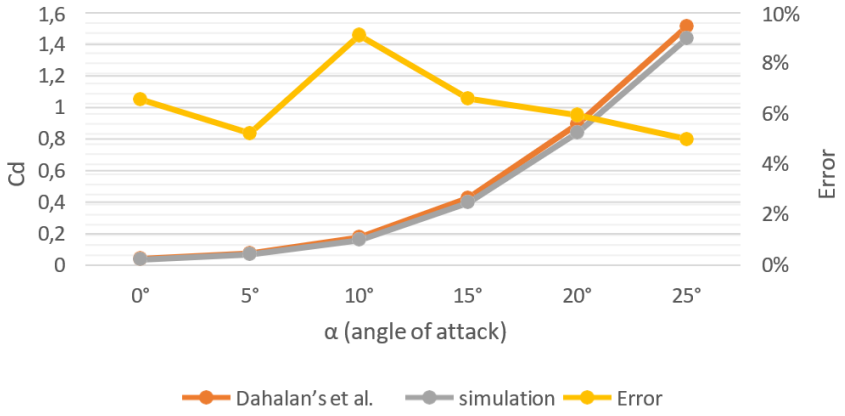


Fig. 15. Previous Work Results by Dahalan

Table 5. Validation Data

AoA (°)	Dahalan et al.	Simulation	Error
0°	0.04054	0.037875127	6.57%
5°	0.07541	0.071464594	5.23%
10°	0.17702	0.160874148	9.12%
15°	0.42713	0.398914966	6.60%
20°	0.89610	0.842762953	5.95%
25°	1.51357	1.437880509	5.00%

The lift coefficient (Cl) for rockets with 3, 4, and 6 fins was obtained from CFD simulations with angles of attack (AoA) ranging from 0° to 25°. The Cl values show a consistent increasing trend as the AoA increases. At 0° AoA, Cl values were 0.00176 for the three-fin configuration, 0.00015 for the four-fin configuration, and 0.0001 for the six-fin configuration, indicating nearly zero lift in the horizontal

position. However, when the AoA increased to 25°, Cl rose significantly to 0.37464 (3 fins), 0.46022 (4 fins), and 0.5257 (6 fins). This increase demonstrates that the number of fins plays an important role in generating lift, due to the greater surface area and the pressure differential between the upper and lower surfaces of the fins. The trend indicates that the six-fin configuration provides the best aerodynamic performance in producing lift compared to the three- and four-fin configurations. The significant rise in Cl is especially evident once the AoA exceeds 10°, which marks the onset of a steep increase for all fin configurations.

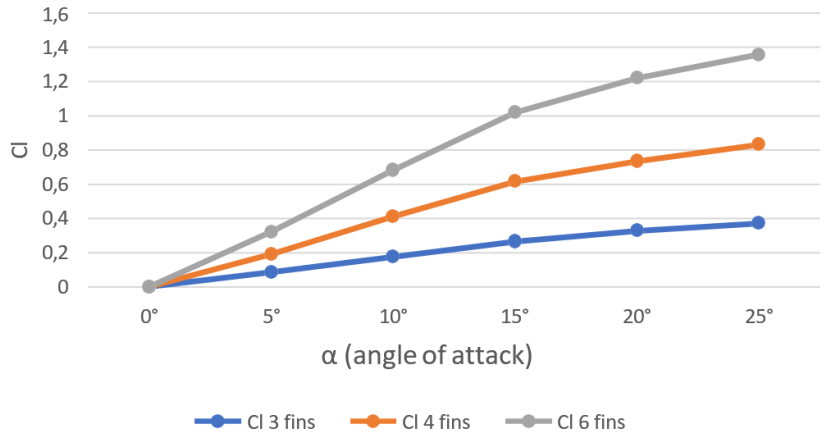


Fig. 16. Lift Coefficient (Cl) Values for Rockets with 3, 4, and 6 Fins

Based on the data and the graph shown in Fig. 16 comparing Cm for rockets with 3, 4, and 6 fins, it can be observed that the pitching moment coefficient (Cm) decreases as the angle of attack (α) increases. The comparison between the number of fins indicates that the six-fin configuration tends to produce the lowest (most negative) Cm values, particularly at a 25° angle of attack, reaching a value of -0.00615.

For the rocket with 3 fins, the Cm values show fluctuations and tend to be higher (less negative) compared to the 4- and 6-fin configurations across most angles of attack. For example, at α = 20°, Cm is -0.0024 and then drops sharply at α = 25° to -0.01665. This indicates that the 3-fin configuration tends to provide better pitching stability at certain angles of attack. In contrast, the 6-fin configuration exhibits more linear and consistent pitching stability as the angle of attack increases.

From an aerodynamic perspective, more negative Cm values generally indicate a stronger nose-down pitching moment, meaning the fin configuration provides a stronger stabilizing effect. Therefore, a higher number of fins, such as six, has the potential to offer better stabilization of the pitching moment at high angles of attack. However, it is also important to balance stability with aerodynamic efficiency to avoid excessive drag.

E-ISSN 2961-8746
P-ISSN 2961-8932

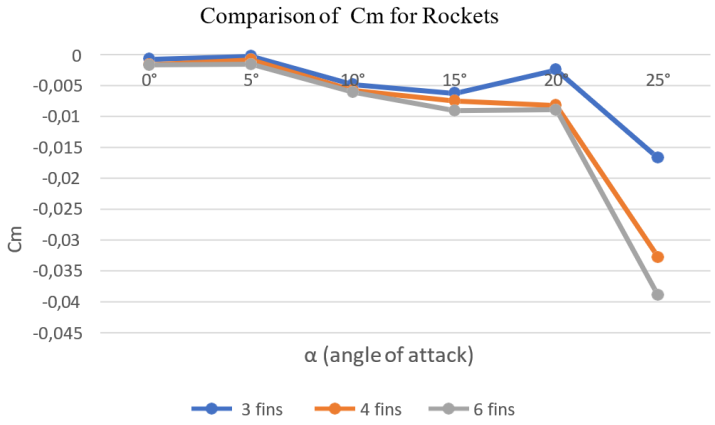


Fig. 17. Coefficient of moments for 3,4, and 6 fins

To determine the rocket fin configuration that provides the best performance, a comparison based on the Cl/Cd ratio is necessary. This is because low Cd and high Cl values individually do not necessarily indicate optimal aerodynamic performance. The Cl/Cd ratio provides a more comprehensive picture of a rocket's aerodynamic efficiency in generating lift with minimal drag.

Based on the simulation results, the highest Cl/Cd ratio is achieved by the rocket with a 6-fin configuration at a speed of Mach 0.6. The graph in Fig. 18 shows that the aerodynamic performance of the 6-fin rocket is consistently better compared to the 3- and 4-fin configurations, particularly at an angle of attack (AoA) of 10 degrees. This indicates that launching the rocket at an AoA of around 10° can achieve the highest aerodynamic efficiency, regardless of the number of fins used. Nevertheless, the 6-fin rocket demonstrates superior performance in terms of both stability and flow pressure distribution, making it suitable for missions that require high stability.

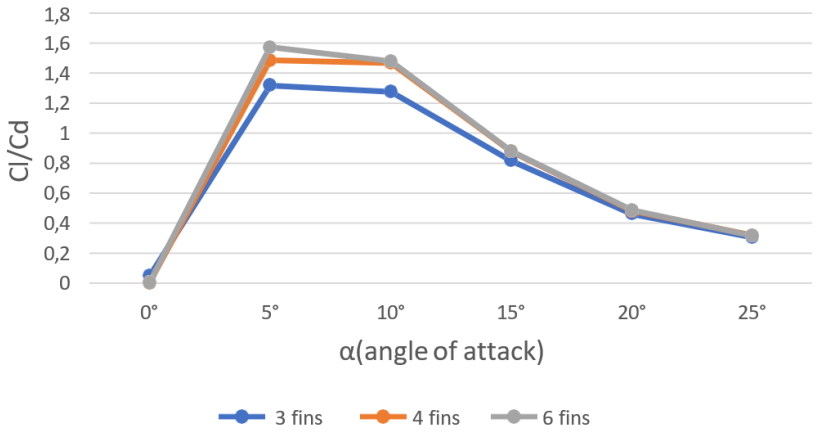


Fig. 18. Comparison of Cl/Cd Values for Rockets with 3, 4, and 6 Fins

Conclusion

In this study, three rocket fin configurations—3 fins, 4 fins, and 6 fins—were analyzed using CFD simulations in the ANSYS environment to evaluate aerodynamic performance at Mach 0.6 and

angles of attack (AoA) ranging from 0° to 25°. The primary parameter observed was the lift-to-drag coefficient ratio (Cl/Cd), which reflects the overall aerodynamic efficiency.

The simulation results show that the rocket with six fins achieves the highest Cl/Cd values, particularly at angles of attack of 5° and 10°, indicating the most optimal aerodynamic efficiency. Increasing the number of fins provides a larger lifting area and a more stable airflow distribution, which is highly beneficial for enhancing lift and maintaining rocket stability. The four-fin configuration exhibits moderate performance, while the three-fin configuration delivers the lowest efficiency.

However, from an economic perspective, a higher number of fins inevitably increases manufacturing costs, total component weight, and assembly complexity. A rocket with six fins requires more material, longer fabrication time, and imposes a greater structural load. Therefore, although this configuration offers maximum aerodynamic benefits, the four-fin configuration could be a more economical and structurally efficient choice, especially for missions that do not require maximum efficiency or are conducted under budget constraints.

Overall, the number of fins has a significant impact on both aerodynamic performance and economic considerations. A rocket with six fins is well-suited for missions that prioritize stability and lift efficiency, while a four-fin configuration offers an optimal balance between performance and cost. Therefore, the choice of fin design should be tailored to the specific mission requirements and available resources.

References

[1] G. P. Sutton and O. Biblarz, *Rocket Propulsion Elements*, 8th ed. Hoboken, NJ, USA: Wiley, 2016.
[2] J. D. Anderson, *Fundamentals of Aerodynamics*, 6th ed. New York, NY, USA: McGraw-Hill Education, 2016.
[3] A. D. N. Barokah, U. Ubaidillah, B. W. Lenggana, et al., "A comparative analysis of aerodynamic performance: Straight fin and curved fin rockets using computational fluid dynamics (CFD)," *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, vol. 115, no. 1, pp. 1–18, 2024, doi: 10.37934/arfmts.115.1.118.
[4] M. Mani and S. Mahjoob, "A comparative investigation into aerodynamic performances of two set finned bodies with circular and non-circular cross sections," *WSEAS Transactions on Fluid Mechanics*, vol. 1, no. 3, pp. 201–207, 2006.
[5] K. Kitamura, K. Fujimoto, and K. Kuzuu, "Numerical analysis on aerodynamic characteristics of advanced solid rocket," *Aerospace Technology Japan*, vol. 10, no. 1, pp. 1–7, 2011, doi: 10.2322/ASTJ.10.1.
[6] J. Sahu, "CFD simulations of a finned projectile with microflaps for flow control," *International Journal of Aerospace Engineering*, vol. 2017, Art. ID 4012731, 2017, doi: 10.1155/2017/4012731.
[7] C. E. De Souza, J. L. F. Azevedo, and F. C. Almeida, "Numerical simulations of the aerodynamic load distributions over the VS-30 sounding rocket," *Journal of Aerospace Engineering*, 2010.
[8] N. Sahbon, M. Michałów, and S. Murpani, "CFD study of base drag of the Grot rocket," *Transactions of the Institute of Aviation*, vol. 247, no. 3, pp. 17–27, 2023, doi: 10.2478/tar-2023-0007.
[9] W. Tang, Y. Xiaofeng, and G. Yewei, "Aerodynamic prediction and performance analysis for Mars science laboratory entry vehicle," *International Journal of Aerospace Engineering*, vol. 9, no. 3, pp. 45–52, 2015.
[10] F. Mandour, K. Mahmoud, and M. Abdalla, "Computational techniques for calculation of missile aerodynamic coefficients," in *Proc. 16th Int. Conf. Aerospace Engineering*, 2010, pp. 1–8, doi: 10.21608/ICEENG.2010.33250.
[11] W. K. Wan Ali and M. H. Shafie, "Simulation study on the effect of asymmetric fin position on aerodynamic characteristic of a missile," *International Journal of Engineering and Applied Sciences*, vol. 7, no. 4, pp. 15–21, 2015.

[12] J. Zhang, H. Liu, and Y. Wang, "Investigation of fin configuration effect on aerodynamic coefficients of rocket at different Mach numbers," *Aerospace Science and Technology*, vol. 43, pp. 178–185, 2015.
[13] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 2nd ed. Harlow, U.K.: Pearson Education, 2007.
[14] M. N. Dahalan, N. H. Nordin, M. A. M. Isa, and N. A. Sidek, "Aerodynamic study of air flow over a curved fin rocket," *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, vol. 45, no. 1, pp. 1–14, 2017.
[15] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA Journal*, vol. 32, no. 8, pp. 1598–1605, 1994, doi: 10.2514/3.12149.
[16] F. M. White, *Viscous Fluid Flow*, 3rd ed. New York, NY, USA: McGraw-Hill, 2006.
[17] A. G. Moulick, "Computational fluid analysis and optimization of rocket engine nozzles at various divergent angles," *Research Square Preprint*, 2023, doi: 10.21203/rs.3.rs-3059286/v1.
[18] C. H.-J. Merrem, V. Wartemann, and T. Eggers, "Preliminary aerodynamic design of a reusable booster flight experiment," *CEAS Space Journal*, vol. 12, no. 3, pp. 337–348, 2020, doi: 10.1007/S12567-020-00313-9.
[19] J. L. Paul, "Aerodynamic optimization of a supersonic bending body projectile by a vector-evaluated genetic algorithm," *Journal of Ballistics Engineering*, vol. 30, no. 4, pp. 223–230, 2016.
[20] J. Lei, Z. Zheng, and C. Huang, "Numerical research of aerodynamic characteristic effects of base jet on supersonic rocket," *Advances in Mechanical Engineering*, vol. 2013, Art. ID 757084, 2013, doi: 10.1155/2013/757084.
[21] N. Sahbon, S. Murpani, and M. Michałów, "A CFD study of the aerodynamic characteristics of Twardowsky and FOK rockets," *Transactions of the Institute of Aviation*, vol. 244, no. 1, pp. 1–12, 2022, doi: 10.2478/tar-2022-0003.
[22] J. Despirito, H. L. Edge, and P. Weinacht, "CFD analysis of a generic missile with grid fins," *AIAA Journal*, vol. 38, no. 5, pp. 911–919, 2000.
[23] P. K. McConnaughey, R. Garcia, and L. W. Griffin, "Computational fluid dynamics applications in rocket propulsion analysis and design," NASA Technical Paper 3452, 1993.

Authors



Mohamad Yamin is the Center for Automotive Research Head at Gunadarma University, Jakarta, Indonesia. His undergraduate degree is in Aeronautics from Institut Teknologi Bandung, Indonesia (1993), and his doctorate (2003) is in Aerodynamics, ILR, TU-Berlin, Germany. His research interests are Computational Fluid Dynamics, Dynamics and Control, EV, Aerodynamics, Energy, and Machine Learning. (email: mohay@staff.gunadarma.ac.id).



Nuril Hadi is a fast-track student majoring in mechanical engineering, Gunadarma University, Depok. Since 2022, he has served as a Laboratory Assistant at center for Automotive Research at Gunadarma University. His research interests include aerodynamics, computational fluid dynamics, and energy conversion. (email: nurilhadi08@gmail.com).