Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 988-996


**RPMME**

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

# A Study On Aerodynamic Shape of Fin of Model Rocket Using Computational Fluid Dynamics (CFD)

**Mohammad Hisham Shahir[1], Azwan Sapit[1*]**

[1]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, Batu Bahat, 86400, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.02.107
Received 30 July 2021; Accepted 02 Dec 2021; Available online 25 December 2021

**Abstract:** The design of the model rocket can affect the model during flight, such as a model will wobble. The model rocket also consists the rocket fins at the bottom of the rocket to provide stability during flight. To avoid the wobble rocket, they must choose the correct shape of the fin. Furthermore, the comparison of the shape of the fin able to generatemore lift force was investigated in this study. Therefore, this study has been carried out inthe computational fluid dynamics (CFD) simulation using ANSYS Fluent. To carry out the simulation, varies 3D shape of the fin has been generated such as rectangle, clipped delta and trapezoid. Then, these 3D shape of the fin has been going through the meshing process. After that, setup the boundary conditions in the ANSYS Fluent. By the end of the simulation, rectangle shows the highest lift coefficient and drag coefficient among the others despite the trapezoid shape has the lowest pressure distribution at upper surface of air foil. In conclusion, this is shown the rectangle shape of the fin has the highest lift force generated and make the rocket can flight to the higher altitude.

**Keywords:** Model Rocket, Fins, Computational Fluid Dynamics, ANSYS, Lift Coefficient

## 1. Introduction

The study of air motion, especially when it is influenced by a solid object, is known as aerodynamics. We can compute the lift forces that allow the model rocket to overcome gravity and drag by studying the movement of air around an object. Since then, there have been many different types of model rockets with various finned shapes. Furthermore, a model rocket can be used as a prototype for constructing a larger rocket. Finned must be fitted to the body of the model rocket to strengthen its

*Corresponding author: azwan@uthm.edu.my

stability. The airflow around the model rocket will be described in this study, as well as the effect of the installation with various finned shapes.

To identify the best shape of fin for the stable design of model rocket. There are a lot of shape of fin and its cause different stability. Therefore, this study to choose the best shapeof fin to reach higher altitude and best aerodynamics to the model rocket. Lastly, the analysis is carried out by using CFD method which is ANSYS.

## 2. Literature Review

### 2.1 Aerodynamic of rocket

Aerodynamic forces are produced and as it moves through the air, they act on a rocket. Forces are vector quantities that have a magnitude as well asa direction. The magnitude of the aerodynamic forces depends on the rocket's form, size and velocity, and on certain characteristics of the air it flies through. The single aerodynamic force is broken into two components by convention: the drag force opposite the direction of motion and the lift force acting perpendicular to the direction of motion. The lift and drag operate through the center of pressure, which is the average position of an object's aerodynamic forces [1-6].

### 2.2 Center of gravity

The center of gravity (CG) is the mass balance point of the rocket, that is, if the rocket was laid horizontal and balanced on a pencil, the CG is the location where the rocket balances [1]. This is important, because this is the point that the rocket would rotate about if it was spun end over end.

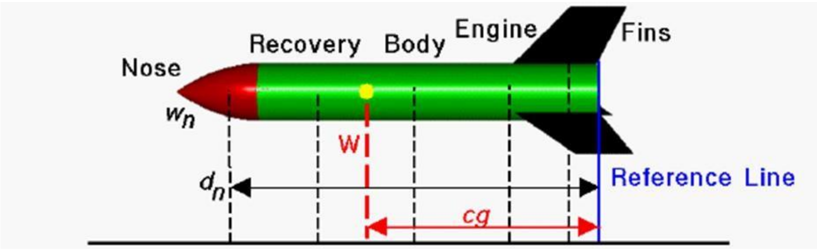



**Figure 1: The center of gravity**

Distance CG times the weight(W) of the rocket sum of the component distance timescomponent weight.

$$CGW = e\ n\ W\ n + e\ r\ W\ r + e\ b\ W\ b + e\ f\ W\ f + e\ g\ W\ g$$

### 2.3 Center of pressure

To produce a stable design, the concept of center of pressure must be utilized. The center of pressure (CP) is that point on a rocket about which the torques generated by aerodynamic forces balance.

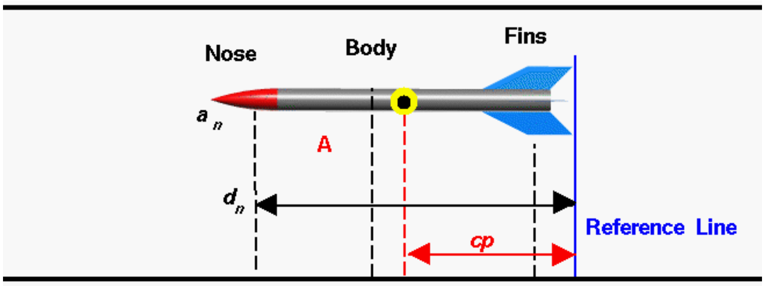



**Figure 2: Center of pressure**

Distance center of pressure times the area, A equals the sum of the component distance area: -

$$CPA = e\,n\,A\,n + e\,g\,A\,g + e\,b\,A\,b$$

### 2.4 Shape of fin

Shape of fin effect of amount of drag and lift force to the rocket [2], stated two things are going on at the same time, and the first thing causes the second thing. The two things are the lift force on the fin is causing a drag force increase on the fin. The core factor is the lift force. The lift is only generated when the symmetrical air foil is oriented on an air foil. 'Angle-of-attack' against the air. That means the fin is tilted in the wind as it flies forward. A rocket needs this lift force to pull the rocket back to a straight path relative to the air flowing over it. In other words, if it were not for the lift force produced by the fins, the rocket would go unstable. Having a lift force to restore the rocket to a straight path is a needed in unguided model rockets. Even though the rocket may not be pointed perfectly straight up, it will still go higher than a rocket that doesn't have any fins. However, the different shape of fin gives a different lift force.

### 2.5 Computational Fluid Dynamics

Computational fluid dynamics is part of the art of replacing the related partial differential fluid flow equations with numbers, and advanced these numbers in space and time to achieve a final numerical definition of the complete flow field of interest [7]. This is not an all-inclusive description of CFD, there are some issues that allow an immediate solution to the flow field without going forward in time or space, and there are some applications that include integral equations rather than partial differential equations. The end of product of CFD is indeed a collection of numbers, in contrast to a closed-form analytical solution. However, in the long run the objective of most engineering analyses, closed form otherwise, is a quantitative description of the problem.

## 3. Methodology

This chapter will clarify the methodology used as part of the research introduced as flow charts. The system was partitioned into two sections: the procedures required before using conventional software and conventional software processes. The conventional software used to simulate fluid flow over the model rocket is ANSYS.

### 3.1 ANSYS FLUENT

The parameter is very important to get the correct and accurate result. After all the parameters are obtained, continue to the next step, analysing the aerodynamic of the model rocket using ANSYS FLUENT. Some five stages need to complete as a procedure.The five stages are geometry modelling, meshing, boundary condition setup, calculation solver, and simulation result.

### 3.2 Geometry Modelling

The geometry modelling of the fin of the model rocket is inspiring by NACA 4415 air foil. This geometry is done in DesignModeler. This geometry of air foil is made by drawing a curve line by plotting XYZ points. The data of XYZ points were taken fromthe website airfoiltools.com. Then, copy the sketch into the new plane and create loft to make 3D geometry. The Boolean feature is utilized to make the wide region have a state of air foil subtracted. This is because the area should have analysed the flow over the air foil only. Then, make a rectangle as a medium to the air while operating the simulation.

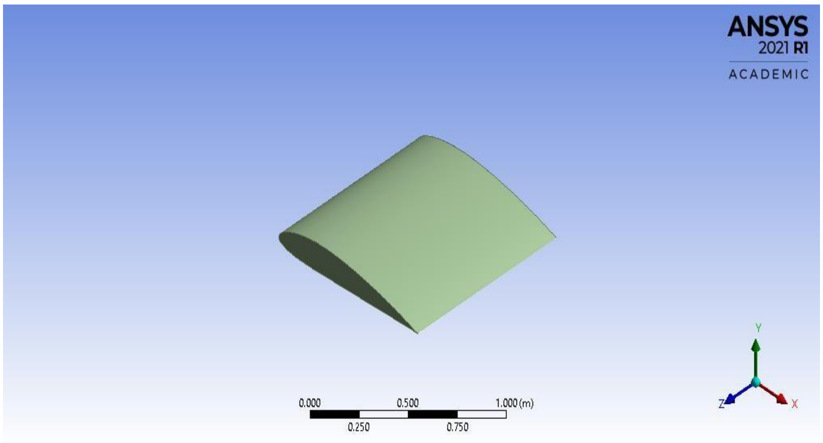



**Figure 3: Rectangle Model**

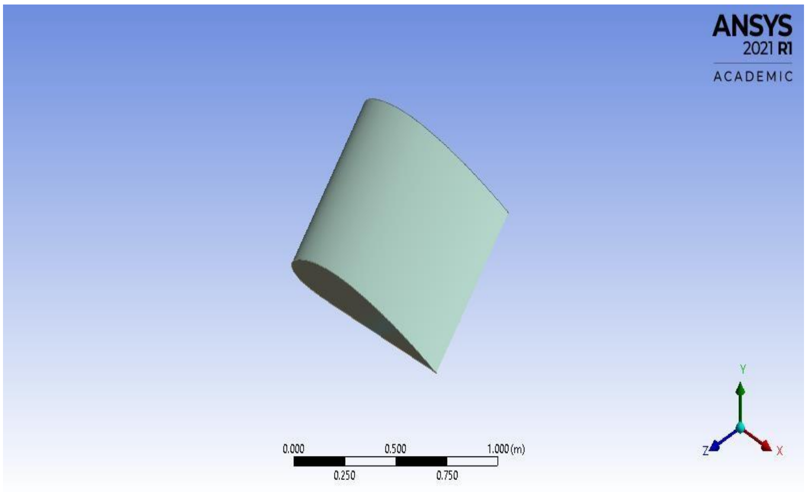



**Figure 4: Clipped Delta model.**

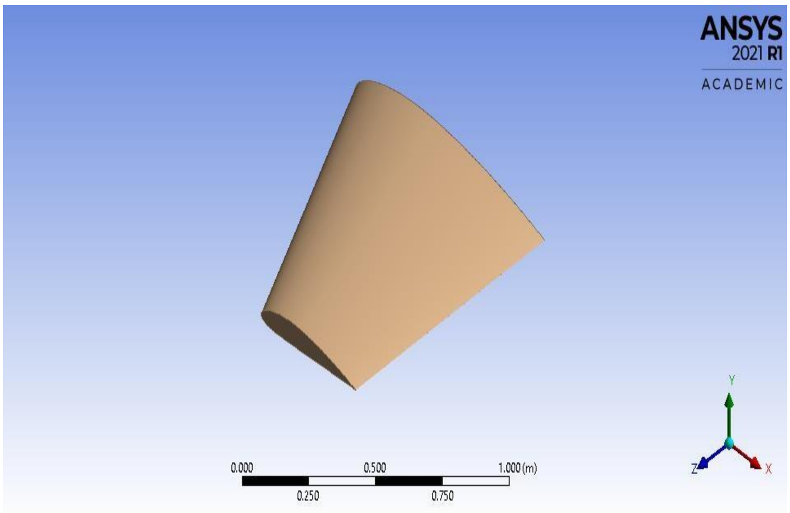



**Figure 5: Trapezoid model.**

## 3.4 Meshing

Meshing is an integral part of the computer-aided engineering (CAE) simulation process. The mesh affects the solution's accuracy, convergence, and speed. Furthermore, the time it takes to construct a Mesh model is often a large portion of the time it takes to get CAE results. As a result, the better the meshing tools and the more automated they are, the better the solution.

| Details of "Mesh" | |
|---|---|
| **Display** | |
| Display Style | Use Geometry Setting |
| **Defaults** | |
| Physics Preference | CFD |
| Solver Preference | Fluent |
| Element Order | Linear |
| Element Size | 7.e-002 m |
| Export Format | Standard |
| Export Preview Surface Mesh | No |
| **Sizing** | |
| Use Adaptive Sizing | No |
| Growth Rate | Default (1.2) |
| Max Size | 7.e-002 m |
| Mesh Defeaturing | Yes |
| Defeature Size | Default (3.5e-004 m) |
| Capture Curvature | Yes |
| Curvature Min Size | Default (7.e-004 m) |
| Curvature Normal Angle | Default (18.0°) |
| Capture Proximity | No |
| Bounding Box Diagonal | 4.9016 m |
| Average Surface Area | 2.7196 $m^2$ |
| Minimum Edge Length | 0.4 m |

| | |
|---|---|
| **Quality** | |
| Check Mesh Quality | Yes, Errors |
| Target Skewness | Default (0.900000) |
| Smoothing | Medium |
| Mesh Metric | None |
| **Inflation** | |
| Use Automatic Inflation | None |
| Inflation Option | Smooth Transition |
| Transition Ratio | 0.272 |
| Maximum Layers | 5 |
| Growth Rate | 1.2 |
| Inflation Algorithm | Pre |
| View Advanced Options | No |
| **Batch Connections** | |
| **Advanced** | |
| Number of CPUs for Parallel Part Meshing | Program Controll... |
| Straight Sided Elements | |
| Rigid Body Behavior | Dimensionally Re... |
| Triangle Surface Mesher | Program Controll... |
| Topology Checking | Yes |
| Pinch Tolerance | Default (4.5e-004... |
| Generate Pinch on Refresh | No |
| Sheet Loop Removal | No |

**Figure 6**: **Details of Meshing**

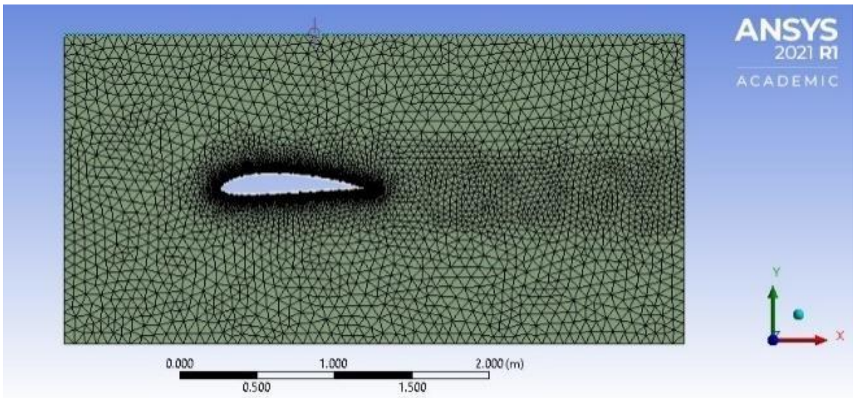



**Figure 7: Meshing of rectangle**

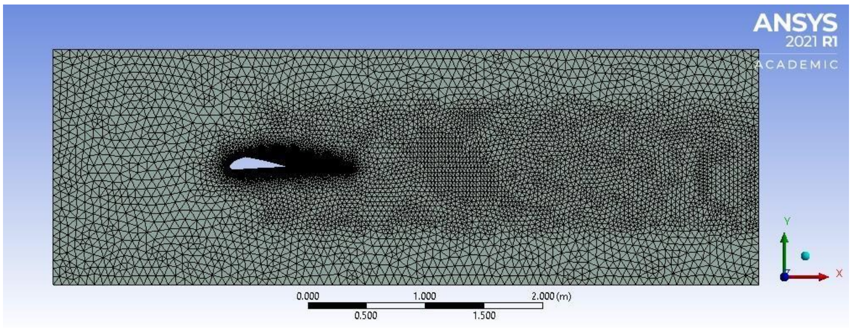



**Figure 8: Meshing of clipped delta.**

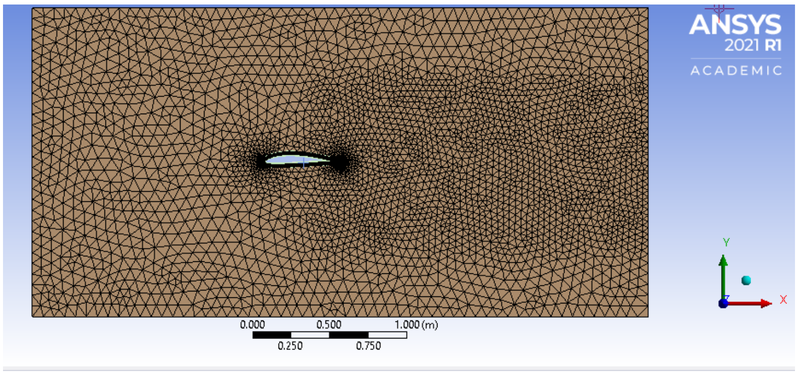



**Figure 9: Meshing of trapezoid.**

3.5 Boundary condition

Boundary conditions have been used extensively in the research. The inlet velocity and outlet pressure are the study work's boundary conditions. Hence, the geometric model is the fin of the model rocket, and the study needs to find the significant inlet velocity and pressure outlet was applied. This indicates that the model has a velocity input and output similar to that of a wind tunnel.

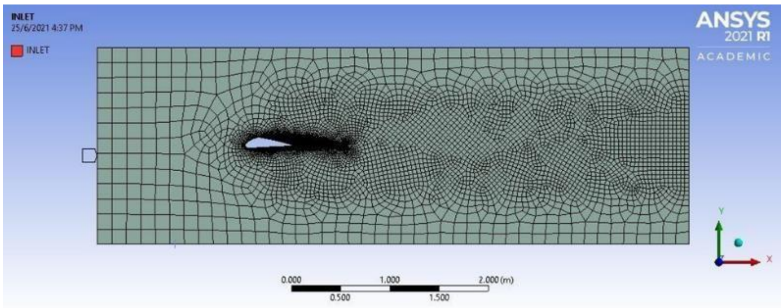



**Figure 10: Velocity inlet**

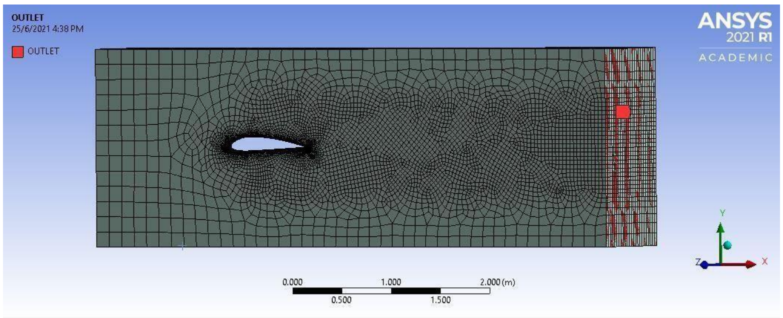



**Figure 11: Pressure outlet**

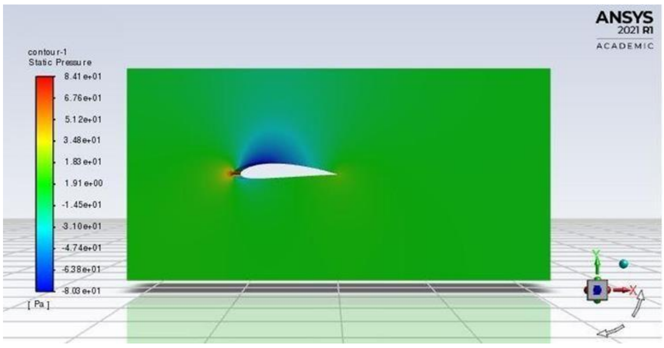
## 4. Results and Discussion

### 4.1 Results simulation

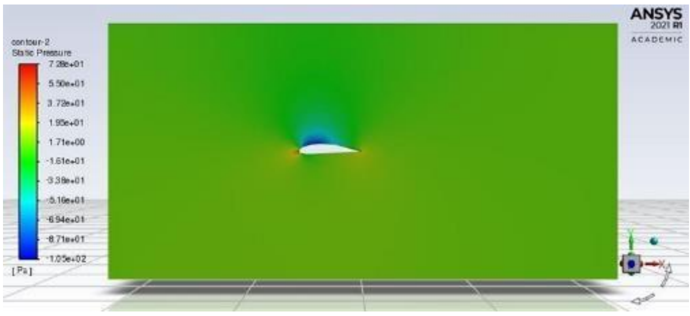
The simulation outcome after the complete iteration is shown in the diagram below. The graphs below depict the results of the analysis.

#### 4.1.1 Contours of static pressure

Static pressure is a term used in fluid dynamics to define the amount of pressure exerted by a fluid that is not moving (Corrosionpedia, 2017). When applying a pressure head, static pressure is commonly measured as a force divided by an area or length.




(a)




(b)

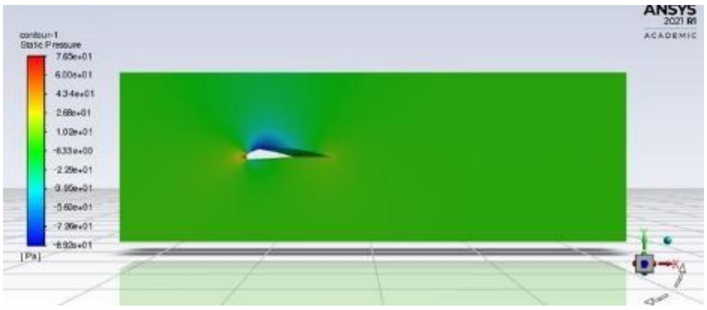
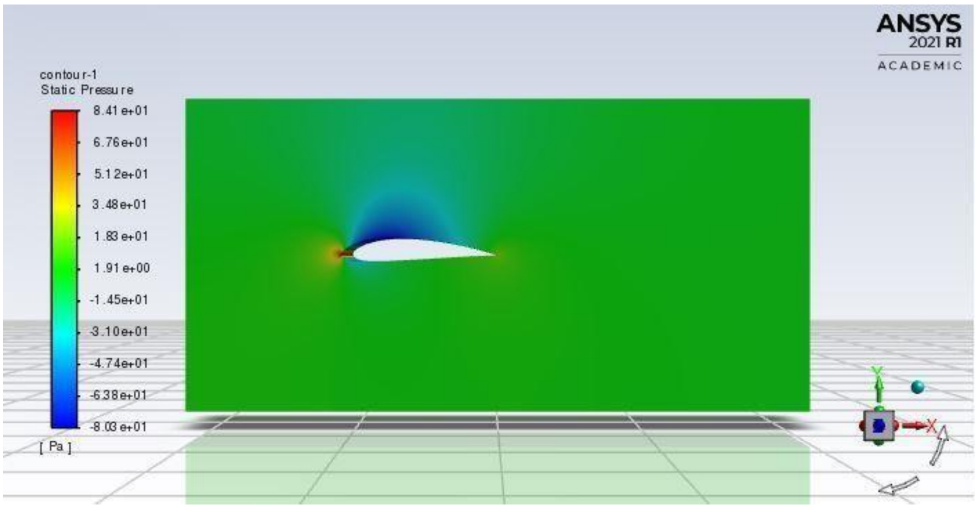



**Figure 12: Contours of static pressure (a) rectangle (b) trapezoid (c) clipped delta**

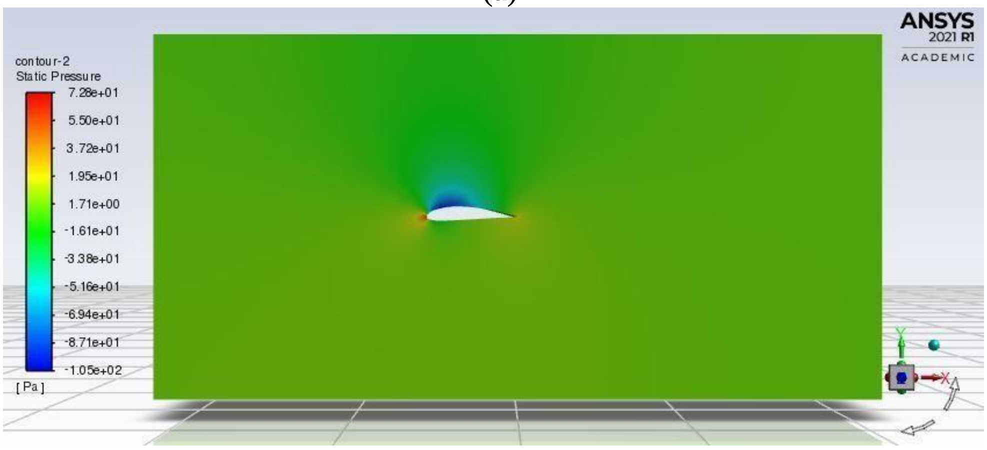
At the leading edge each shape has the red contour of static pressure, and this is due to the high pressure at the stagnation point. The pressure the rectangle has the most significant area of static pressure contours, but the pressure is not the lowest. Meanwhile, the trapezoid has the smallest area of static pressure contours but has the lowest pressure. As expected, all shapes produce a lift force significantly.

4.1.2 Contours of velocity magnitude

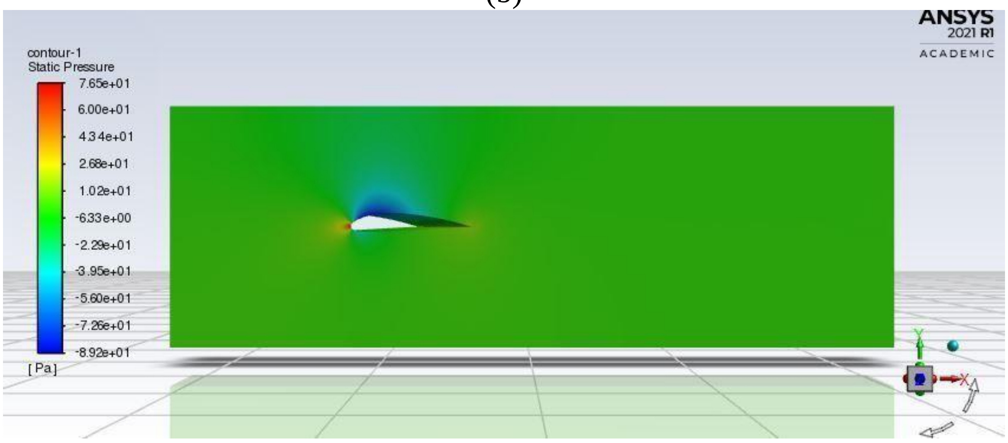
As expected from the nature of pressure distribution, higher velocity is experienced in theupper surface compare to the lower surface. Velocity at the upper surface is increased than the lower surface of the air foil. Low velocity at a lower surface generates more lift.




(a)




(b)




(c)

**Figure 13: Contours of magnitude velocity (a) rectangle (b) trapezoid (c) clipped delta**

4.2 Lift Coefficient and Drag coefficient

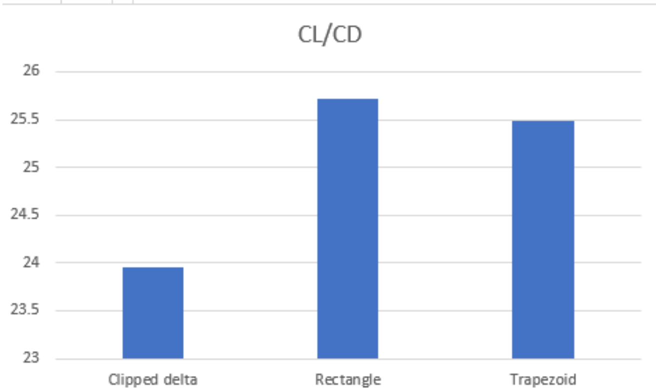



**Figure 14: The graph shows CL/CD vs shape of fin.**

It is obviously showing that lift coefficient and drag coefficient for the rectangle is the highest at once cause the lift force higher than other shape of the fin. Therefore, CL/CD for the rectangle is the highest too. These cause the rectangle fin generate more lift rather than others. In addition, higher lift force causes the center of pressure move downwards. Since that, it makes the model of rocket can flight in stable condition.

## 5. Conclusion

The main objective of this study has achieved, to compare the shape of the fin of the model rocket and stimulate by using CFD. The air foil is from NACA4415 and developed into 3D modelling and shaped into rectangle, clipped delta and trapezoid. In conclusion, the lift coefficient and drag coefficient was generated by the end ofthe simulation. The results show clearly that the rectangle has the largest lift coefficient and drag coefficient, resulting in a larger lift force than alternative fin shapes.

**Acknowledgement**

The authors wish to thank to the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia that has supported on the accomplishment of research activity.

## References

[1] Richard Nakka's Experimental Rocketry Site. (2021). Retrieved July 3, 2021, from Nakka- rocketry.net website: https://www.nakka-rocketry.net/fins.html

[2] Tim Van Milligan. (2017). What is the best fin shape for a model rocket? Retrieved from https://apogeerockets.com/education/downloads/Newsletter442.pdf

[3] Cronvich, L.(n.d.). Missile Aerodynamics. Retrieved from website: https://www.jhuapl.edu/Content/techdigest/pdf/V04-N03/04-03-Cronvich.pdf

[4] Price, D. (1957). Free-Body Modeling Of The Dynamics Of A Fin- Stabilized Ballistic Missile In Nonspinning Vertical Trajectories. Retrieved from website: Https://authors.library.caltech.edu/57817/1/E-73.1.pdf

[5] William L. Oberkampf, John D. (1971). Aerodynamics of Finned Missiles at High Angle of Attack. AIAA Journal, 9(12), 2378–2384. doi:10.2514/3.50043 doi.org/10.2514/3.50043

[6] In IJRET: International Journal of Research in Engineering and Technology (pp. 2321–7308).

Retrieved from website: https://ijret.org/volumes/2015v04/i03/IJRET20150403016.pdf

[7] Rose John, B. R., Sivathanu, B. U., & Vijaya, B. B. J. (2018). Computational Investigation on the Effects of Two-Way Wing Morphing Strategy in the Critical Mach Number. Journal of Aerospace Technology and Management, 10. https://doi.org/10.5028/jatm.v10.825