Stress and CFD Analysis of the aviation sector Wings for Better Performance: A Numerical Study

Mahesh [1], Ravi M Tilavalli [2], G Veerabhadrappa [3]

[1] *Department of Mechanical Engineering, Government polytechnic Aurad, Karnataka, India.*

[2] *Department of Mechanical Engineering, Government polytechnic Harihara, Karnataka, India.*

[3] *Department of Mechanical Engineering, Government polytechnic Karatagi, Karnataka, India.*

ABSTRACT

One of the most essential components in the construction of an aircraft wing is the air foil section, which may be thought of as the embodiment of a wing or a lifting surface. Despite the fact that the geometry of the air foil is changing, the aerodynamic properties of the air foil are also changing. The purpose of this inquiry is to examine the influence of changes in form that occur as a result of modest differences in the coordinates. The reference for this study is a conventional symmetrical air foil. Following the completion of this optimization procedure, three novel designs of air foil have been generated. The findings of the aerodynamic characteristics, such as the coefficients of lift and drag (C_d and C_l), the pressure coefficient (C_p), and the moment coefficient (C_m), are recorded for each of the three distinct profiles that were generated from the standard NACA 0012.Furthermore, our proposal incorporates the use of air foils manufactured by Wortmann fx 63-137 and Clark y. Within the context of this optimization process, the Computational Fluid Dynamics (CFD) method serves as the modus operandi. ANSYS FLUENT and MODAL have been used by us in order to do flow and stress analysis. The flow changes that have occurred for these different forms of air foil have been documented, and the findings have been arrived at in order to determine which air foil is the most suitable.

Keywords: Airfoil, airfoil shape, aerodynamic characteristics, Fluent, CFD, Mach number.

1. INTRODUCTION

Due to the fact that a fixed-wing aircraft is unable to fly without the wing, it is possible that the wing is the most crucial component of an aircraft. The structure is composed of an airfoil that has a certain amount of cross sectional area. We start the process of detail design by designing the wings since the geometry of the wings and the characteristics of the wings have an effect on all of the other components of the aircraft. One of the most important functions of the wing is to provide an adequate amount of lift force, often known as lift (L). Nevertheless, the wing is responsible for two additional products, which are the drag force, often known as drag (D), and the nose-down pitching moment (M). A wing designer's primary objective is to increase lift, but they must also decrease drag and pitching moment in order to achieve optimal performance. It is a common misconception that a wing is a lifting surface. In reality, lift is generated by the pressure differential that exists between the bottom and upper surfaces of the wing. When it comes to the camber line, the thickness of a wing that is following the laminar stream is much greater in the center. This indicates that there is a negative weight gradient along the stream. Therefore, if we keep the camber in the middle, we will be able to generate a laminar stream that has a high rate of flow at a high speed.

2. WING DESIGN

Due to the fact that a fixed-wing aircraft is unable to fly without the wing, it is possible that the wing is the most crucial component of an aircraft. We start the process of detail design by designing the wings since the geometry of the wings and the characteristics of the wings have an effect on all of the other components of the aircraft. The fundamental purpose of the wing is to provide an adequate amount of lift force, often known as lift (L). Nevertheless, the wing is responsible for two additional products, which are the drag force, often known as drag (D), and the nose-down pitching moment (M). A wing designer's primary objective is to increase lift, but they must also decrease drag and pitching moment in order to achieve optimal performance. It is a common misconception that a wing is a lifting surface. In reality, lift is generated by the pressure differential that exists between the bottom and upper surfaces of the wing. The texts on aerodynamics may be reviewed in order to refresh your memory on the mathematical procedures that are used to compute the pressure distribution across the wing and how to find the flow variables. In essence, the process of designing wings is carried out in accordance with the ideas and methodology of "systems engineering in its entirety." Limiting variables in the wing design approach derive from design criteria such as performance requirements, stability and control requirements, producibility requirements, operational requirements, cost, and flight safety requirements. Also included in this category are flight safety requirements. Stall speed, maximum speed, take-off run, range, and endurance are some of the most important performance criteria at this point. There are three primary needs for stability and control: lateral-directional static stability, lateral-directional dynamic stability, and the ability to maneuver the aircraft regardless of whether or not the wings are likely to stall. For the purpose of calculating wing lift, wing drag, and wing pitching moment, an aerodynamic method is one of the instruments that is required throughout the process of designing flying wings. In light of the advancements that have been made in the field of aerodynamics, there is now a wide range of methods and instruments available to complete this laborious task. Throughout the course of the last several decades, a wide range of tools and software that are based on aerodynamics and numerical approaches have been produced. The computational fluid dynamics (CFD) software that is now available on the market is based on the solution of Navier-Stokes equations, the vortex lattice technique, the thin air foil theory, and circulation. At this early point in the design process of the wings, it seems that the use of such software, which is both very costly and time-consuming, is not required. An alternative, more straightforward method known as Lifting Line Theory is presented instead. By using this theory, one is able to calculate the production of those three wings (L, D, and M) with a degree of precision that is satisfactory.

3. Methodology

CFD and CAE are examples of numerical strategies that are used in the process of reproducing physical problems via the utilization of experiments. This allows for inquiries on the arrangement to be made without the need to create a physical model. In a broad sense, the following procedures are included in the process of numerical simulation.

Modeling: Following the end of the aerofoil selection process, we gathered aerofoil coordinate data from the website of aerofoil tools in accordance with the measurements that were necessary. The coordinates are imported into the CREO program, which then allows for the generation of a two-dimensional model. A 3D model is then created by extruding the 2D model that was formed as shown in figure. When the various varieties of aerofoils have been modeled, the data is then loaded into ANSYS WORKBENCH for computational fluid dynamics (CFD) and MODAL analysis. For the purpose of analyzing the different properties of the aerofoils, even more software known as XFLR is used. Figures following illustrate the three-dimensional model of NACA 0012 that was created using the CREO program, as well as the mesh of one of the aerofoils that was chosen.

Fig 1: 3D model of NACA Aerofoil 0012

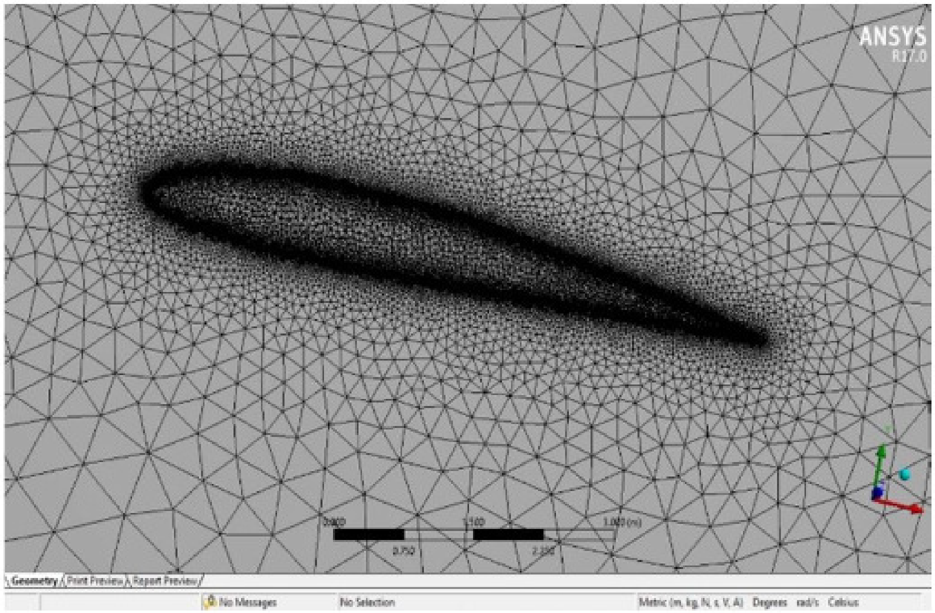

Fig 2: NACA 0040 Mesh part

Meshing: The default mesher in Ansys was used to mesh each and every one of the distinct aerofoils. For the purpose of capturing the boundary layers, a minimum of five inflation layers were produced around the aerofoils. When we gave it a fine mesh, we were able to create the best mesh around the aerofoils, which is seen in the image shown above. We can see the structure of the mesh elements of the different aerofoils in the table that is provided below.

Table 1: No. of nodes and elements of different aerofoils

AIRFOIL	NO. OF NODES	NO. OF ELEMENTS	MESH TYPE
NACA 0012	192663	353937	Triangular
NACA 0015	194248	353441	Triangular
NACA 0030	198568	361220	Triangular
NACA 0040	201320	387756	Triangular
Clark Y	326001	593100	Triangular
Wortmann fx63-137	248640	400882	Triangular

Fluent: The aerofoils were imported into Ansys once they had been properly meshed via the process. In order to get the simulation started, both the fluid and boundary conditions were provided. All of the various aerofoils were taken into consideration in order to produce a variety of pressure and velocity contours. Following is a list of the numerous boundary criteria that were implemented in this situation.

Table 2: Boundary conditions used in Fluent

S.No	PROPERTIES	VALUES
1	Type of material	Aluminium
2	Chord	300mm
3	Span	1m
4	Velocity	133 m/s
5	Solution method	Standard k-ε

XFLR5: Following the acquisition of the several contours of pressure and velocity, C_l and C_d values were calculated with the assistance of a piece of software known as XFLR5. We are able to readily get the graphs of C_l and C_d for a variety of aerofoils inside this program, and we are able to compare them in order to come up with the L/D ratio.

Modal Analysis: Ansys Workbench was used once again to import the aerofoils after they had been successfully iterated in Fluent. The purpose of this import was to determine the natural properties of the aerofoil, also known as the mode shape and vibration frequencies, which are responsible for the vibratory oscillations of the wind turbine. The self-weight condition was chosen in this instance for the primary purpose of determining the flutter characteristics of the aerofoil profiles and the possible causes of noise-related issues that occurred throughout the flight. It is even possible to predict the structural performance of the aerofoil if one is aware of the mechanisms of failure.

Comparison: Following the collection of all the output data from the different software programs, all of the aerofoils were compared in order to identify the optimal aerofoil that is capable of providing superior performance and stability.

4. RESULT ANAYSIS

It is possible to see the various contours of velocity and pressure distribution on various aerofoils by referring to the figures that are provided below. It can be observed quite plainly that the pressure and velocity of each aerofoil are different from one another.

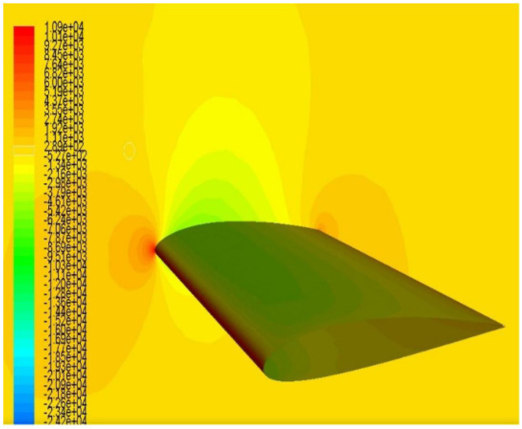

Fig 3: Pressure distributions on NACA 0030

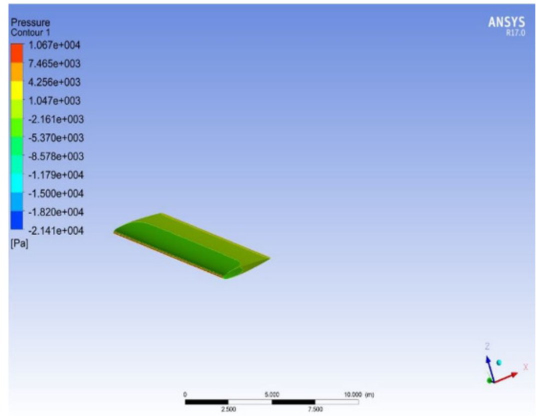

Fig 4: Pressure distribution on Wortmann fx63-137

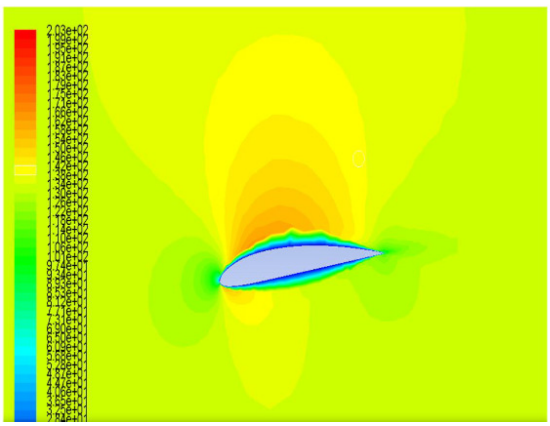

Fig 5:Velocity distribution of Wortmann fx63-137

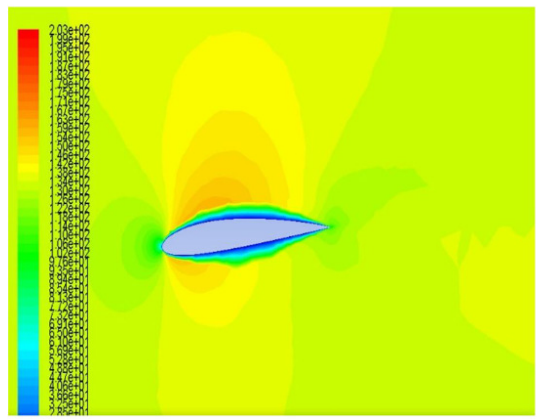

Fig 6:Velocity distribution of NACA 0040

Following the discovery of the various contours of velocity and pressure, we have taken a variety of aerofoils and computed the C_l, C_d, and C_m values for each of those aerofoils. Following that, a graph was created that compared those aerofoils, and the L/D ratio was also determined based on the data received. After conducting an investigation of airfoil forms under a variety of situations, including Mach number and angle of attack, it was discovered that shape had an influence on the lift-to-drag ratio. The results of this analysis are given below. Table 3 displays a selection of the tabulated data that were obtained.

The aerodynamic coefficients are influenced not only by the body shape of the aircraft (the airfoil section that is selected), but also by the attitude (angle of attack), Reynolds number, Mach number, surface roughness, and air turbulence associated with the aircraft. When the angle of attack is set to three degrees, the results are reported in Table 5.2, which displays the (L/D) ratio of the Clark Y and Wortmann airfoils.

Table 4: (L/D) ratio in % of chord of NACA 0012

Mach No.	REF	1	2	3
	(L/D) in % of chord at AOA 3°			
	12	15	30	40
0.2	21.9	15.5	23.9	30.3
0.3	21.0	15.0	23.2	29.4
0.4	20.5	14.5	22.6	28.5
0.5	20.5	14.0	22.0	27.1
0.6	19.0	12.9	21.6	27.1

Table 5: (L/D) ratio in % of chord

Mach no.	Wortmann f*63-137	Clark y
	(L/D) in % of chord at 3 AOA	
0.2	61.7	41.5
0.3	60.5	40.3
0.4	59.2	39.1
0.5	58.1	38.0
0.6	56.0	37.5

The design and analysis of the wing that was created in the XFLR5 program are shown in the graphics below. It was necessary to apply boundary conditions such as the Mach number, the Angle of Attack, the chord length, and the dihedral angle.

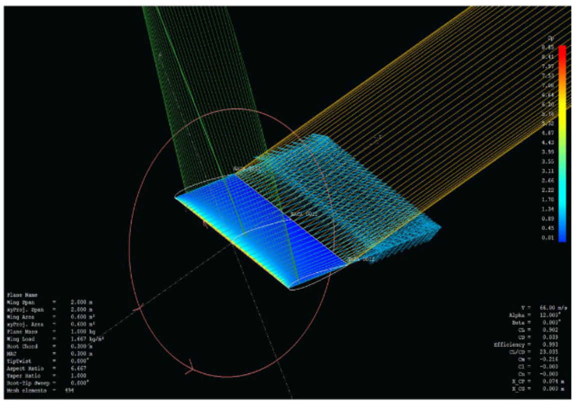

Fig 7: XFLR5 model of NACA 0012

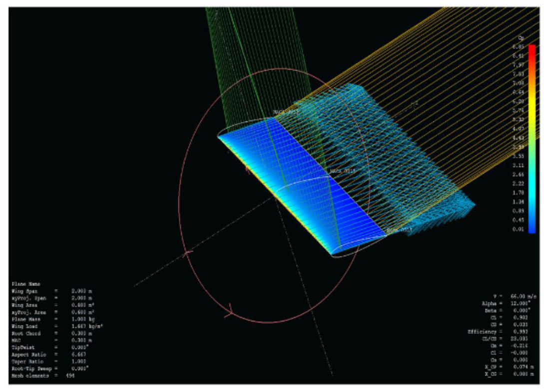

Fig 8: XFLR5 model of NACA 0015

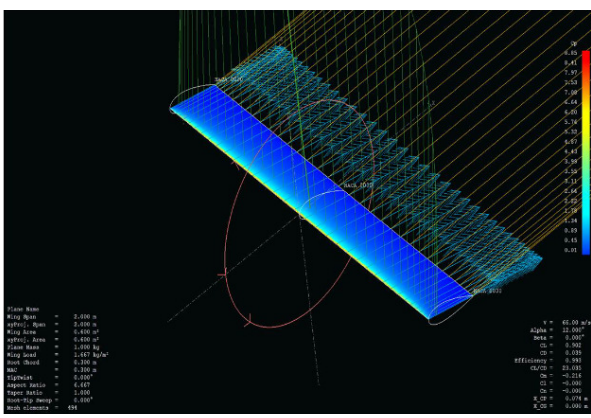

Fig 9: XFLR5 model of NACA 0030

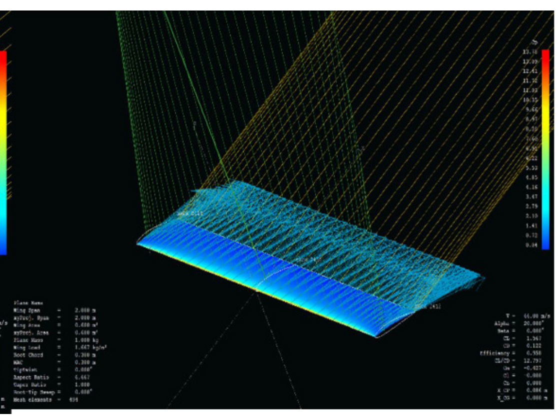

Fig 10: XFLR5 model of NACA 0040

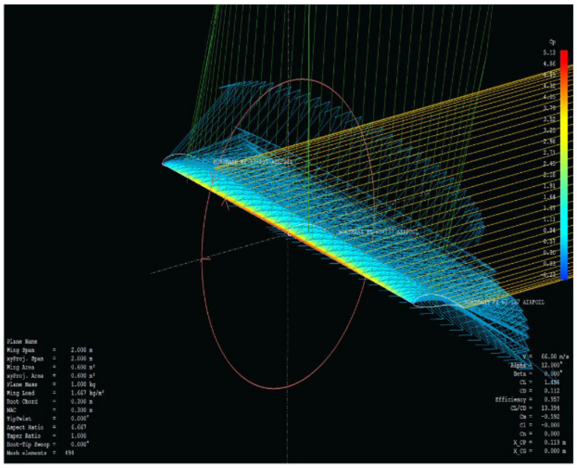

Fig 11: XFLR5 model of Wortmann fx63-137

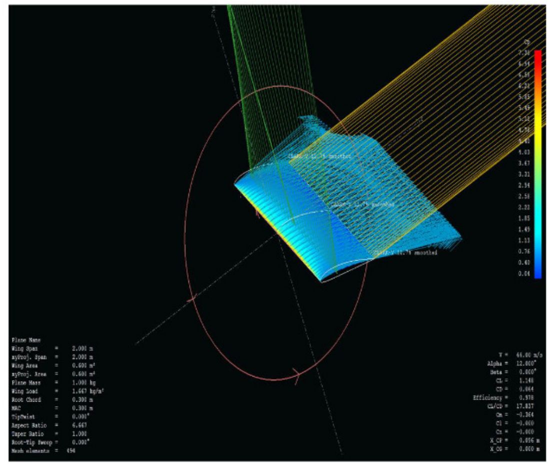

Fig 12: XFLR5 model of Clark Y airfoil

The following figures shows the stress distribution on each aerofil and the toatl deformation in them.

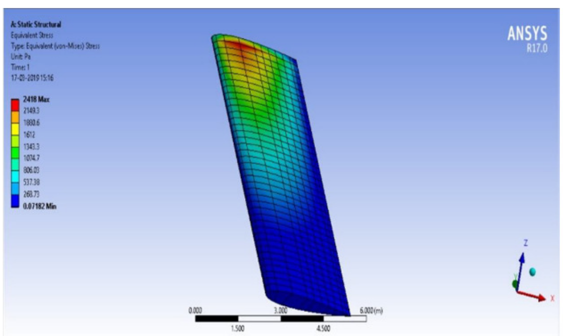

Fig 13: Stress distribution of NACA 0012

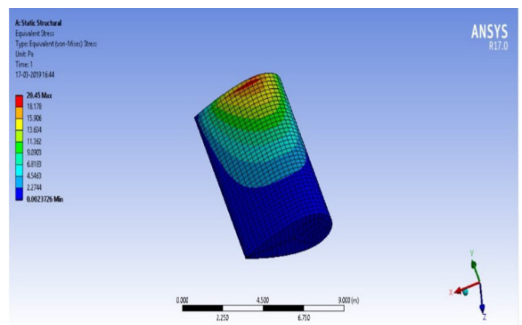

Fig 14: Stress distribution of NACA 0030

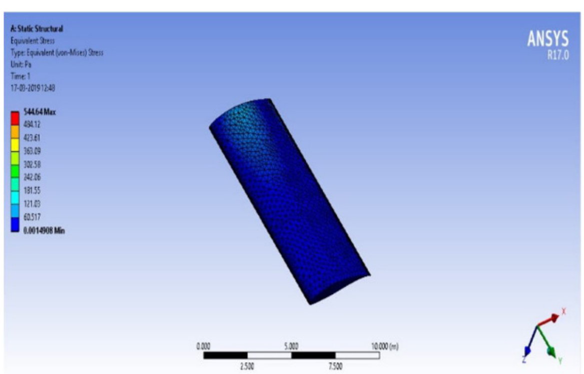

Fig 15: Stress distribution of Wortmann fx63-137

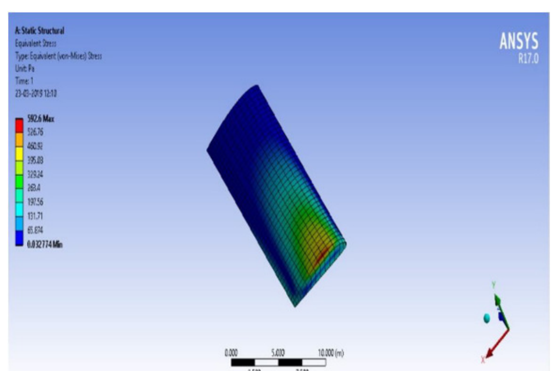

Fig 16: Stress distribution of Clark Y

5. Conclusions

In light of the findings of the study, the following assessments have been made. CFD and MODAL analysis were performed on a wide variety of aerofoils, each of which had a varied thickness expressed as a percentage of chord. In terms of lift to drag values, it was discovered that the Wortmann fx63-137 has the optimum values when the angle of attack is 12 degrees and the Mach number ranges from 0.2 to 0.6. According to the findings that were acquired using ANSYS FLUENT, it has been discovered that the wortmann fx63-137 has the most favorable pressure and velocity distributions among the many airfoils that were considered.With the use of the XFLR5 program, we created graphs that contrasted Cl with Cd, Cl/Cd with α, and Cm with α. Figure illustrates the comparison between a number of different airfoils. It has been discovered that Wortmann fx63-137 has the maximum Cl value when the Cd level is low. Based on the results of the modal analysis, the material wortmann and clark y has the highest resistance to stretching and stresses.

References

1. Jithendra Sai raja chada, Sri ram deepak akella et. Al : Study Of Flow Characteristics In a Convergent And Divergent Nozzle Using Computational Fluid Dynamics International Journal of Scientific Research and Engineering Development-– Volume 4 Issue 3, May -June 2015.
2. K. M. Pandey, S. K.Yadav : CFD Analysis of a Rocket Nozzle with Four Inlets at Mach 2.1 International Journal of Chemical Engineering and Applications, Vol. 1, No. 4, December 2010 ISSN: 2010-0221.
3. K. M. Pandey and Virendra Kumar : CFD Analysis of Four Jet Flow at Mach 1.74 with Fluent Software International Journal of Chemical Engineering and Applications, Vol. 1, No. 4, December 2010 ISSN: 2010-0221.
4. Kousik kumaar. R, kesavan.M : Design and cfd analysis of shock waves over supersonic cd nozzle international journal of latest trends in engineering and technology (IJLTET) ISSN:2278-621X, p-502 to510.
5. Arun Kurien Reji, G. Kumaresan, Archit S Menon, Jithesh Parappadi, Harikrishna A. P, Ajaykrishnan Mukundan International Journal of Pure and Applied Mathematics Volume 119 No. 12 2016, 2135-2142.
6. H Pujowidodo, A I Siswantara2, G G R Gunadi. : A Daryus The Study of Converging-Diverging Nozzle for Improving the Impulse Momentum of Cross-Flow Turbine in a Bio-Micro Power Plant ICB2018 IOP Conf. Series: Earth and Environmental Science 209 (2018) 012054 IOP Publishing doi:10.1088/1755-1315/209/1/01205.
7. Gamble, Dwain Terrell, P.E. Rich DeFrancesco : Nozzle selection and design criteria AIAA-2004-3923.
8. E. M. S. Ekanayake, J. A. Gear, Y. : Ding Flow simulation of a two dimensional rectangular supersonic convergent-divergent nozzle ANZIAM J. 51 (EMAC2009) (page number C377–C392), 2010.

9. Mhammad Arif Hussain, M.Satya Prasad, Dr. S.Nagakalyan : Design and flow analysis of various convergent - divergent nozzles by using computational fluid dynamics (CFD)-International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163 Issue 05, Volume 6 (page number 372 to 379) doi://10.26562/IJIRAE.2019.MYAE10087.
10. Bogdan-Alexandru Belega, Trung Duc Nguyen : Analysis of flow in convergent-divergent rocket engine nozzle using computational fluid dynamics -international conference OF scientific paper afases Brasov, 28-30 May 2015.
11. Raghu Ande, Venkata N. Kumar Yerraboina Numerical Investigation on Effect of Divergent Angle in Convergent-Divergent Rocket Engine Nozzle-chemical engineering transactions vol. 66, 2018 from (page number 787 to 792) DOI: 10.3303/CET1866132.
12. Uhammad Arif Budiyanto, Jansen Novri, Muhammad Idrus Alhamid, Ardiyansyah : Analysis of Convergent and Divergent-Convergent Nozzle of Waterjet Propulsion by CFD Simulation-AIP Conference Proceedings 2062, 020066.
13. E. M. S. Ekanayake, J. A. Gear, Y. : Ding Flow simulation of a two dimensional rectangular supersonic convergent-divergent nozzle ANZIAM J. 51 (EMAC2009) (page number C377–C392), 2010.
14. Ambareen Khan, Abdul Aabid, S. A. Khan : CFD analysis of convergent-divergent nozzle flow and base pressure control using micro-JETS International Journal of Engineering & Technology, 7 (3.29) (2016) (page number 232-235).
15. M.H.M. Noh, A.H.A. Hamid, R. Atan, And H. Rashid Numerical investigation of chocked converging-diverging nozzles for thruster applicationiium Engineering Journal, Special Issue, Mechanical Engineering, 2011(p no- 10 to 18).
16. V. M. Krushnarao Kotteda· Sanjay Mittal : Flow in a planar convergent-divergent nozzle Article in Shock Waves.
17. Madhu B.P, Syed Sameer, Kalyana Kumar M, Mahendra Mani G : CFD Analysis of convergent-divergent and contour nozzle. International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 8, August 2016, pp. 670–677, Article ID: IJMET_08_08_073.
18. Kyle J. Krogh : Developing a Framework for Control of Agile Aircraft Platforms in Autonomous Hover. Master of Science in Aeronautics & Astronautics, University of Washington, 2009, “YAK 54 Dimensions”