International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 04 Issue: 12 | Dec -2020 ISSN: 2582-3930

Numerical Study on Flow Separation and Angle of attack using CFD

Nitin Kumar Saidha, Student, Department of Mechanical Engineering, Vellore institute of technology

Abstract— This work involves usage of Reynolds Averaged Navier Stokes Equation (RANSE) based Computational Fluid Dynamics (CFD) approach for the determination of the relation between flow separation and angle of attack on NACA Boeing 737-IL Outboard Airfoil. Geometrical modelling of this aerofoil has been done using STAR ccm+ and simulations have been carried out using the same software.

Keywords: Airfoil; Flow separation ; Angle of attack; Coefficient of drag; Coefficient of lift;

1. INTRODUCTION

In this study we used a Boeing 737-IL Outboard airfoil to study flow separation. An airfoil is defined as the cross section that is placed in an airstream in order to produce an effective aerodynamic or hydrodynamic force in the most efficient manner possible. The cross sections of wings, canards, propeller blades, windmill blades, and blades in a jet engine are all examples of airfoils. It is a fact that a body in motion through a fluid experiences a resultant force mainly a resistance to a motion. When an airfoil body passes through any fluid it produces an aerodynamic force which is due to pressure distribution over the body surface and shear stress distribution over the body surface. This aerodynamic force can be resolved into two components known as lift and drag. The force which acts in perpendicular to the direction of motion is called lift force, and the force which is parallel to the direction of motion is called drag force. Lift generated by aerofoil primarily depends upon surface area, geometry of airfoil and angle of attack. Stall is a sudden reduction in the lift generated by an airfoil as angle of attack increases. This occurs when the angle of attack exceeds the critical angle. The critical angle of attack is usually about 15-20 degrees, but it may vary significantly depending on the fluid, airfoil, and Reynolds number. [3] ,[4] ,[5]

On reviewing various literatures, it can be understood that quite a lot of research has been done to find the angle at which stall occurs, prevent stall et al. Thus the idea of this work is to establish a relation between flow separation before stall conditions.

2. COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics is the part of fluid mechanics that investigates issues including liquid stream by utilizing mathematical examination, numerical analysis and information structures. The communication of fluid and gases with a surface characterized by limit conditions are addressed with the assistance of computers, which plays out its iterations. CFD is commonly accepted as referring to the subject encompassing the numerical solution, by numerical methods, of the equations which describe fluid flow, the set of Navier-stokes equations, continuity and any additional conservation equation like species concentration or energy equations.. CFD is considered as a middle ground between physical experimentation and pure theory. Computational fluid dynamics predict the performance of a system before creating it in real life. CFD predicts which design changes are most crucial to enhance the performance. Moreover, there are several unique advantages of CFD over experimental-based fluid system design**:**

- A great time reduction and cost reduction in new designs
- There is a possibility to analyze different problem whose experiments are very difficult and dangerous
- The CFD techniques offer the capacity of studying systems under conditions over their limits.
- The level of detail and accuracy is practically limitless.

CFD Analysis of temperature, velocity, and chemical Concentration distribution can help an engineer to understand the problem correctly and provide ideas for getting the best resolution. The CFD software used for this study is Siemens' Star CCM+ software.

3. FLOW SEPARATION AND STALL

Whenever an object travels through a fluid it experiences viscous forces, this occurs in the layer of fluids close to the solid surface which acquires a boundary layer of fluid around it. Boundary layers can be in the form of laminar or turbulent. By calculating the Reynolds number of the fluid flow conditions one could make a decision about the characteristics of the flow (whether the flow is laminar or turbulent). Flowing against a rising pressure is known as flowing in an adverse pressure gradient. The boundary layer separates when it has travelled far enough in an adverse pressure gradient that the speed of the boundary layer relative to the surface has stopped and reversed direction.[6] This happens because the boundary layer travels far enough against an adverse pressure gradient and at the same time the speed of the boundary layer relative to the object falls almost to zero, this is when flow separation occurs. Flow separation can often result in increased drag and reduced lift. For this reason, much effort and research have gone into the design of aerodynamic surfaces which delay flow separation and keep the flow attached for as steep as possible.

4. GEOMETRY

Airfoil geometry can be characterized by the coordinates of the upper and lower surface. It is often summarized by a few parameters such as: maximum thickness, maximum camber, position of max thickness, position of max camber, and nose radius. For this study a Boeing 737 outboard airfoil from NACA has been used. Though there are a lot of airfoil geometries and understanding their basic characteristics is important. Subsonic airfoils have a round leading edge around which it is naturally insensitive to angle of attack. The point in front of the airfoil that has maximum curvature is known as a Leading edge. The trailing edge is defined as the point
of minimum curvature at the rear of the airfoil. The the straight line joining the leading edge and trailing edge of the airfoil section is the chord line. Camber line is the locus of point's midway between the upper and lower surface of an airfoil. The ratio of a span of an aerofoil to the chord length of an airfoil called the Aspect ratio.

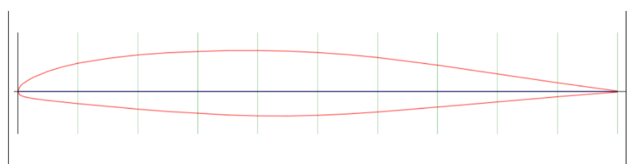

Fig1: Geometry of Selected Airfoil

5. SIMULATION SETUP

5.1GEOMETRIC MODELLING

First all coordinates of the airfoil are taken from NACA's website. Airfoil in 2 dimensions for smooth or symmetric was prepared in CAD software for further investigation. A Wind tunnel with symmetry planes as shown in fig.2 was prepared to analyse the airfoil's behaviour. A wind tunnel with symmetry planes works as a wind tunnel having larger dimensions than airfoil in such a way that its edges do not affect the flow and results.

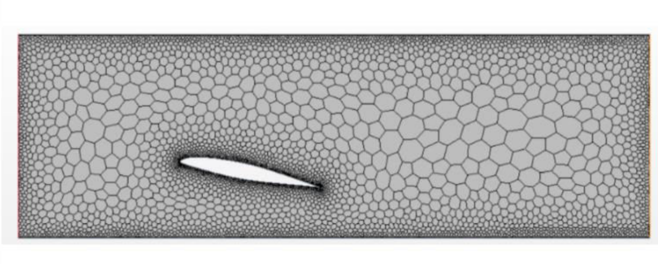

Fig2: Geometric Mesh

5.3 OPERATING PARAMETERS

Analysis on the airfoil profile is carried out to find the values of CD and CL at different values of angle of attack .Here we are going to analyse the Airfoil of Chord Length 160 mm using CFD. For that we take some initial assumptions or boundary conditions for our problem which are as follows.

Sl No	Parameter	Value
1	Velocity	33.33m/s
2	Density	1.208Kg/m3
3	Airfoil Length	0.16m
4	Fluid	Air (Ideal)
5	Operating Pressure	13 bar
6	Turbulent Viscosity Ratio	10
7	Viscous Model	Spalart Allmaras
8	Momentum Model	Second order upwind

5.4 Pressure Probe

An effective way of measuring flow separation is to measure pressure above the airfoil. The Probe is placed just above the boundary layer of the airfoil and equidistant to the trailing and leading edge. The pressure is calculated at that point for every angle of attack from 0-20 degrees. (The placement of the probe is shown in Fig3)

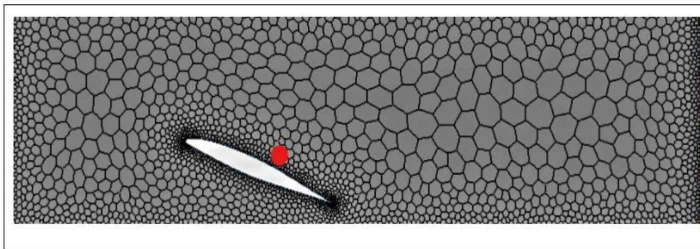

Fig3. Placement of Pressure Probe

6. RESULTS AND DISCUSSION

Simulations for various angles of attack were done in order to be able to compare the results from the different turbulence models and then validate them with existing experimental data from reliable sources. To do so, the model was solved with a range of different angles of
attack from 0 to 20°. On an airfoil, the resultants of the forces are usually resolved into two forces and one moment. The component of the net force acting normal to the incoming
A flow stream is known as the lift force and the component of the net force acting parallel to the incoming flow stream is known as the drag force. The curves of the lift and the drag coefficient are shown for various angles of Attack.

6.1 Visualisation of Lift and Drag

Fig 4-6 Visually represent the Data (Magnitude of Velocity) on the given airfoil. From the contours, we see that there is a region of low velocity or high pressure at the leading edge (stagnation point) and a region of low pressure and high velocity on the upper surface of airfoil. (This is because from the Bernoulli equation, we know that whenever there is high velocity, we have low pressure and vice versa.) Fig 4 and 6 have similar velocity profiles with a gradual (and expected) velocity difference between the top and the bottom of the airfoil, however there is a huge drop in velocity above the airfoil at 16°. This is due to “stall” or separation of air flow from the boundary. At this point the flowstream velocity of air above the airfoil is relatively low.

Volume: 04 Issue: 12 | Dec -2020 ISSN: 2582-3930

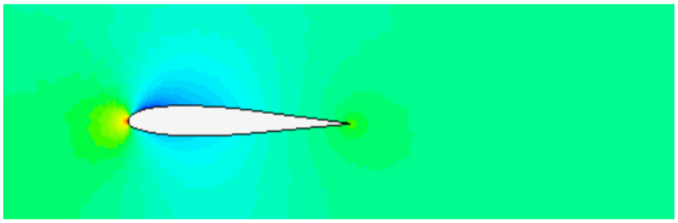

Fig4. Velocity magnitude Profile at 0° angle of attack

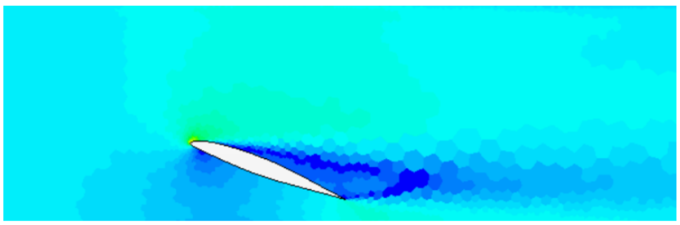

Fig5. Velocity magnitude Profile at 16° angle of attack

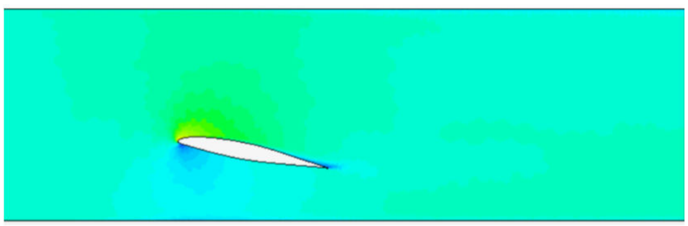

Fig6. Velocity magnitude Profile at 8° angle of attack

6.2 Graphical representation of Collected Data

The data collected from the simulation is plotted into 3 Graphs. Lift and Drag Force vs Angle of attack (Smoothed)(Fig 7), Lift: Drag vs Angle of attack (Fig8) and Pressure at Probe vs Angle of attack. All these graphs show similar characteristics, there is a severe drop or rise in characteristics at 16°. The reason for this must be due to stall (Hence 16° is the critical Angle). This also shows that the Flow separates from the airfoil at a consistent rate until the stall angle is achieved at which point the flow separates entirely. The pressure probe shows a consistent decrease in pressure on the top of the airfoil before the critical angle, this is a crucial observation.

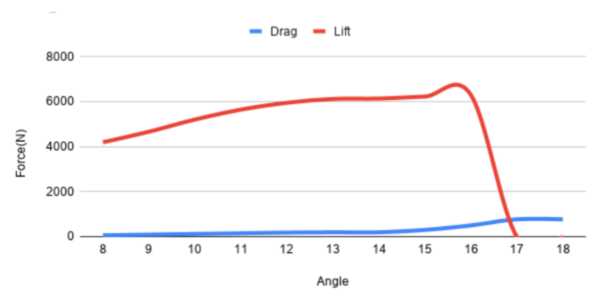

Fig7.

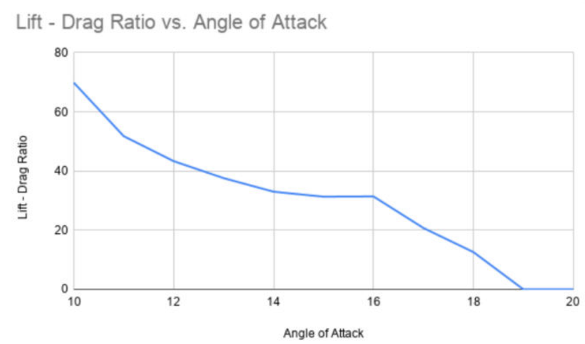

Fig8.

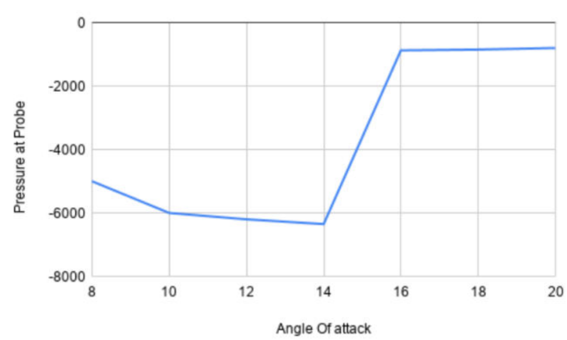

Fig9.

7. Conclusion

In the simulation performed, it was observed that the pressure above the airfoil continues to become more negative as the angle of attack of the airfoil increases until the critical angle is met. This data is concomitant with the Drag/Lift Data. As the pressure becomes more negative, higher lift force is generated. This is also in line with visually represented data such as streamlines and the velocity profile of the wing that show lower velocities at the bottom of the airfoil.

The Visual representation also shows that there is a partial separation of the stream at the top of the airfoil at angles of attack which exceed 8°. This can be inferred from the low velocity spots shown above the airfoil.
This Data when recapitulated allowed us to conclude:

1. CFD provides an accessible platform to understand flow separation and associated phenomena around airfoil bodies.
2. Any (partial) flow separation that happens before the critical angle does not affect the lift or drag. This is because it is overcome by the decrease in velocity below the airfoil, thereby creating the pressure difference needed.
3. Studies should focus on flow separation before stall conditions to increase the efficiency of airfoils and creating a more linear relation between Angle of attack and Lift produced.
4. After the stall angle, the flow separation occurs away from the trailing edge which reduces the lift generated. The critical angle for the given airfoil under the conditions above is found to be 16°.

8. References

[1]Daniel P. Raymer, Aircraft Design: A Conceptual Approach. 3rd edition. AIAA Education Series, Reston, VA, 1999, pp. 39.
[2] Robert J. McGhee, William D. Beasley, and Dan M. Somers, "Low Speed Aerodynamic Characteristics of a 13-Percent-Thick Airfoil Section designed for General Aviation Applications," NASA TM X-72697, NASA Langley Research Center, Hampton, Virginia.
[3] Harris C. D., "Two-dimensional aerodynamic characteristics of the NACA0012 airfoil in the Langley 8-foot transonic pressure tunnel". NASA TM-81927; 1981.
[4] Peter M. Goorjian, Guru P. Guruswamy, "Transonic unsteady aerodynamic and aero elastic calculations about airfoils and wings", Computers & Structures, Volume 30, Issue 4, 1988, Pages 929-936.
[5] Ünver Kaynak, Jolens Flores, "Advances in the computation of transonic separated flows over finite wings", Computers & Fluids, Volume 17, Issue 2, 1989, Pages 313-332.
[6] Anderson, John D. (2004), Introduction to Flight, Section 4.20 (5th edition)
[7]Ashish Kadve, Dr. Prashant Sharma, and Abhishek Patel. "Review on CFD Analysis on aerodynamic design optimization of wind turbine rotor blade", International Journal of Innovation and Emerging Research in Engineering, February 2016, Volume 3, Issue 5, PP 178-183.