A Comparative Study of Boeing 737 and NACA 2412 Airfoils using CFD

Sudip Chakraborty [1]
Assistant Professor,
Department of Mechanical Engineering
ADAMAS University
BARASAT, INDIA

Subhadip Bhattacharjee [2]
Undergraduate Student,
Department of Mechanical Engineering
JIS College of Engineering
KALYANI, INDIA

Saroj Karmakar [3]
Undergraduate Student,
Department of Mechanical Engineering
JIS College of Engineering
KALYANI, INDIA

Susanta Sadhukhan [4]
Undergraduate Student,
Department of Mechanical Engineering
Sarojmohan Institute of Technology
HOOGLY, INDIA

Mr. Arijit Mukherjee [5]
Assistant Professor,
Department of Mechanical Engineering
ADAMAS University
BARASAT, INDIA

Dr. Ranjan Kumar [6]
Assistant Professor,
Department of Mechanical Engineering
ADAMAS University
BARASAT, INDIA

***Abstract*— In this work flow analysis of two airfoils (NACA 2412 and Boeing 737) was investigated. By varying angle of attack, change in drag force and lift force were also analyzed. The outcome of this investigation was shown and computed by using ANSYS workbench 19.0. The pressure distribution as well as change in lift and drag force of these two aerofoils were visualized and compared. From this result, we compared the better aerofoil between these two aerofoils. The whole analysis is solely based on the principle of finite element method and computational fluid dynamics (CFD).**

***Keywords*— NACA 2412, Boeing 737, CFD, ANSYS.**

I. INTRODUCTION

Aerodynamics is the study of how air interacts with moving bodies. It is primarily concern with the forces of drag and lift, which are caused by air passing over and around solid bodies. The flow of air over the aerofoils is the most important thing that has to be considered during designing an aircraft, missile, sport vehicles or any other aerodynamic objects. Computational fluid dynamics (**CFD**) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computational fluid dynamics provides a qualitative and sometimes even quantitative prediction of fluid flow by means of mathematical modeling, numerical method and software tools. ANSYS is vast computational software that enables researchers to analyze the problems related to different engineering sectors. ANSYS Mechanical finite element analysis software is used to simulate computer models of structures, electronics, or machine components for analyzing strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, turbulence, industrial machineries, explicit dynamics and other attributes.

Mustafa Yilmaz et. al.,*" A Comparative CFD analysis of NACA 0012 and NACA 4412 airfoils**"**, they had investigated that 7^0 angle of attack is the optimum value for NACA 0012 and 6^0 angle of attack is the optimum value for NACA 4412.[1] ***Liyana Kharulaman et. al.," Research on Flows for NACA 2412 Airfoil using Computational Fluid Dynamics Method***",*** they observed that the velocity magnitude for compressible flow was much higher compared to incompressible flow.[2] ***Douvi C. Eleni et. al.,"*** Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil***",*** they observed that the computational grid split in two regions, a laminar and a turbulent region.[3] ***Mohamed AboZeed Mohamed Atiah et. al.,"*** Boeing 737-700NG Airplane Flow Simulation***",*** they demonstrate the aerodynamic characteristics of the aircraft.[4] ***MD. Safayet Hossain et.al.,"*** A COMPARATIVE FLOW ANALYSIS OF NACA 6409 AND NACA 4412***",*** They investigate that the NACA 4412 is better than NACA 6409 in terms of less wake generation and less negative pressure on the upper surface of the aerofoil for same angle of attack.[5] ***Mohamed A. Fouad Kandil et.al.,"*** Performance of GOE-387 Airfoil Using CFD***",*** they observed that the magnitude of the aerodynamic moment remains nearly constant when the area.[7] ***Ovais Gulzar et.al.,"*** Impact of Variation in Angle of Attack on NACA 7420 Airfoil in Transonic Compressible Flow Using Spalart-Allmaras Turbulence Model ***",*** they observed that the 40 angle of attack has been found suitable as the pressure, velocity, mach number encountered are within safe limits for an subsonic flow.[8]

In this study, pressure distribution was analysed along with lift and drag force of two aerofoils (NACA 2412 and Boeing

737) by varying angle of attack and velocity. Later, lift and drag force was compared between these aerofoils to find out the suitable one.

angle of attack changes.[6] ***Mragank Pratap Singh et. al.,"*** CFD Simulation of an Airfoil at different Angle of Attack"**,** they concluded that lift and drag force also depend on the wing planform and on the wing Theoritical Analysis

The lift phenomenon can be explained by using Bernoulli's equation. Bernoulli's principle states that an increase in speed must accompany any reduction in pressure; and a speed decrease must accompany any pressure increase. When the air passes over the aerofoil, velocity increases as the air continues to flow from its leading edge to the upper surface of the aerofoil and the pressure is decreased in that area. But on the other hand, velocity decreases as the air passes through the bottom of the aerofoil and the pressure is increased. This positive pressure acting upward acts as the key ingredient for generating lift.

Wake can be defined as a region of flow trailing the body where the effects of the body on velocity are felt. Wake consists of vortices which are responsible for creating drag by creating negative pressure in that region. Wake can occur in an aerodynamic body having a large angle of attack (more than 150 for maximum aerofoil). Separation of boundary layer also depends upon the Reynolds Number, if the number is more, the flow exhibits early transaction from laminar to turbulent.

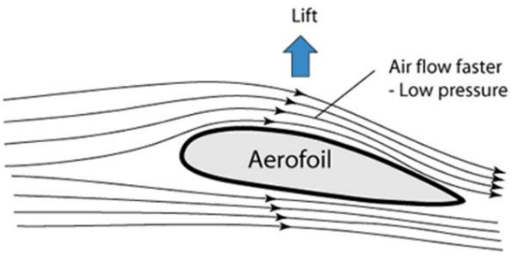

Fig. 1(a). Lift force generation

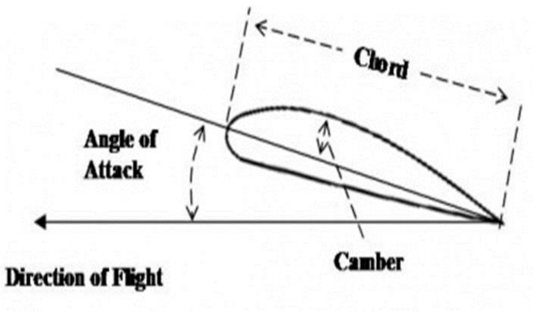

Fig. 1(b). Lift, Drag and Angle of Attack on the Airfoil

Reynolds number R_e is defined by,

$$R_e = \frac{\rho U x}{\mu} \quad (1)$$

Here x is the thickness of boundary layer where the transition from laminar to turbulent starts.

II. ANALYSIS

For carrying out the simulation ANSYS '19.2 has been used and the steps involving into research work are as follows

A. GEOMETRY

In the following figures (i.e. Fig. 2 & 3), NACA2412 and Boeing 737 profiles are shown as 2d sketch, respectively. The coordinates of the airfoils were taken from airfoil database. The coordinates of the airfoil were imported to ANSYS Workbench and to create the geometry of the airfoils.

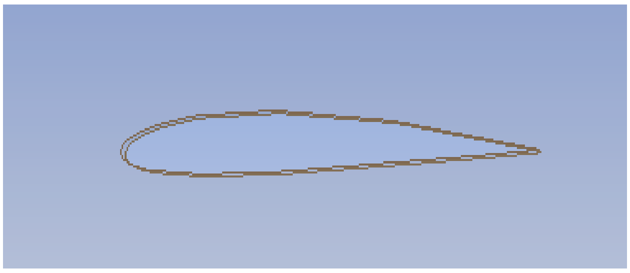

Fig. 2. Geometry of NACA 2412

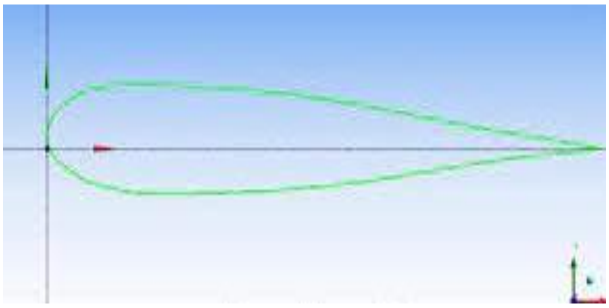

Fig. 3. Geometry of Boeing 737

B. MESH GENERATION

In our study, it is important to catch wakes in leading edge and trailing edge. Therefore, to obtain reliable resolution after trailing edge, it had better attain tight cells in terms of mesh size. Four different analyses have been done by taking tetrahedron methods and maximum number of elements which has been taken is 1683040.

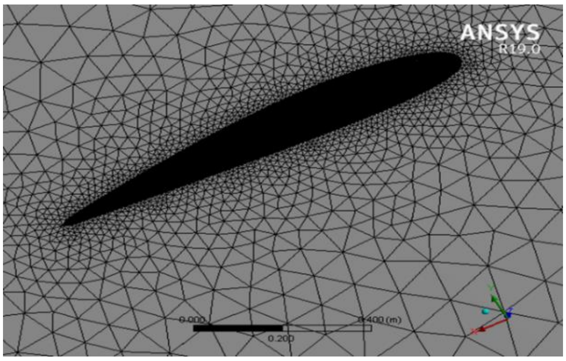

Fig. 4. Mesh generation

C. INITIAL INLET CONDITION

The analysis consists of flow around an airfoil at various angles of attack (-5, 0, 5, 10, 15, 50, 70 degree). The inputs and boundary conditions are shown in the table below

Table 1. Initial Inlet conditions for simulation

Number of iterations for the solution	500
Turbulence Model	RNG k-epsilon model, SST k-omega model
Air Density (ρ)	1.225 kg/m^3
Dynamic Viscosity (μ)	1.789 x 10^{-5} kg/m-sec
Initial Inlet Velocity (InVel)	125 m/sec
Initial Angle of Attack (AoA)	5 degree

D. SIMULATION RESULTS

The simulation results of NACA 2412 and Boeing 737 has shown below –

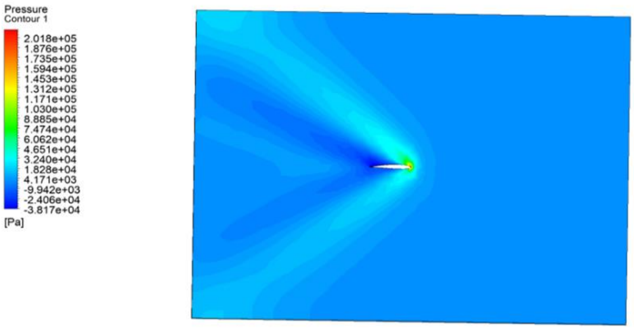

Fig. 5. Pressure Contour

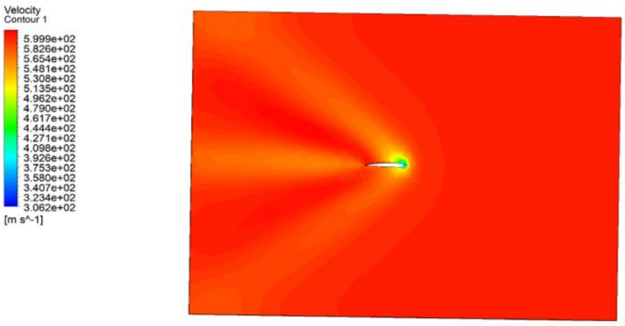

Fig. 6. Velocity Contour

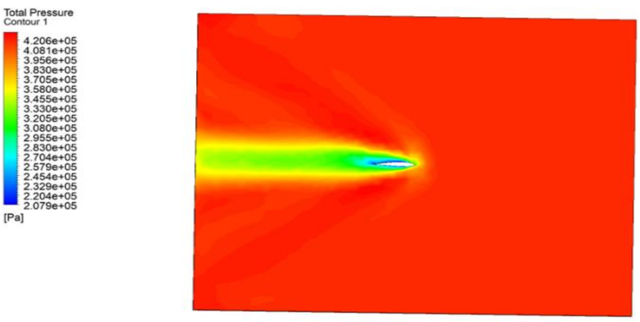

Fig. 7. Total Pressure Contour

1) SIMULATION RESULTS FOR NACA 2412 RNG K-EPSILON MODEL ANALYSIS

Table 2. Variation of lift and drag force with angle of attack keeping inlet velocity = 125 m/sec

Angle of Attack (AoA) (Degree)	Lift Force (N)	Drag Force (N)
-5	-3034.4	94.296
0	3220.6	72.70
5	8524.8	182.69
10	11338	549.09
15	17288	3547.2
50	50357	48874
70	39174	72347

Table 3. Variation of lift and drag force with inlet velocity keeping angle of attack = 0 degree

Inlet Velocity (InVel) (m/sec)	Lift Force (N)	Drag Force (N)
100	2052.6	48.061
125	3220.6	72.698
150	4658.4	102.06
175	6435.0	136.42
200	8325.2	174.09
225	10202	221.83
250	12570	266.61

2) SIMULATION RESULTS FOR NACA SST K-OMEGA MODEL ANALYSIS

Table 4. Variation of lift and drag force with angle of attack keeping inlet velocity = 125 m/sec

Angle of Attack (AoA) (Degree)	Lift Force (N)	Drag Force (N)
-5	3058.9	98.341
0	3191.7	76.334
5	8489.6	183.81
10	10859	601.47
15	15467	2490.6
50	50278	48623
70	39296	72501

Table 5. Variation of lift and drag force with inlet velocity keeping angle of attack = 0 degree

Inlet Velocity (InVel) (m/sec)	Lift Force (N)	Drag Force (N)
100	2018.5	50.703
125	3199.1	76.56
150	4636.9	107.37
175	6396.2	142.64
200	8289.1	182.16
225	10107	234.1
250	12423	279.84

3) SIMULATION RESULTS FOR BOEING 737 RNG K-EPSILON MODEL ANALYSIS

Table 6. Variation of lift and drag force with angle of attack keeping inlet velocity = 125 m/sec

Angle of Attack (AoA) (Degree)	Lift Force (N)	Drag Force (N)
-5	-4281.5	2938.3
0	-1711.6	2617.7
5	12925	3579.6
10	34410	6152
15	57513	8262.4
50	$1.34*10^5$	$1.2*10^5$
70	79768	$1.632*10^5$

Table 7. Variation of lift and drag force with inlet velocity keeping angle of attack = 0 degree

Inlet Velocity (InVel) (m/sec)	Lift Force (N)	Drag Force (N)
100	-808.39	1687.7
125	-1169.4	2629.2
150	-1754.4	3772.0
175	-2489	5117.9
200	-3325.1	6667.7
225	-4252.4	8421.6
250	-5274.4	10379

4) SIMULATION RESULTS FOR BOEING 737 SST K-OMEGA MODEL ANALYSIS

Table 8. Variation of lift and drag force with angle of attack keeping inlet velocity = 125 m/sec

Angle of Attack (AoA) (Degree)	Lift Force (N)	Drag Force (N)
-5	-2450.6	2975.3
0	-1840.4	2628.3
5	11836	3506.8
10	34604	5884.3
15	57921	8303.9
50	$1.31*10^5$	$1.2*10^5$
70	79859	$1.63*10^5$

Table 9. Variation of lift and drag force with inlet velocity keeping angle of attack = 0 degree

Inlet Velocity (InVel) (m/sec)	Lift Force (N)	Drag Force (N)
100	-691.45	1703.3
125	-1840.4	2628.3
150	-3144.8	3757.7
175	-3456.8	5122.6
200	-3801.9	6689.7
225	1822.5	8591.1
250	5827.3	10642

III. RESULT AND CONCLUSION

➢ FOR NACA 2412 AEROFOIL USING RNG K-EPSILON AND SST K-OMEGA MODEL, THE VARIATIONS OF LIFT AND DRAG FORCE WITH ANGLE OF ATTACK AND INLET VELOCITY ARE REPRESENTED USING GRAPHS.

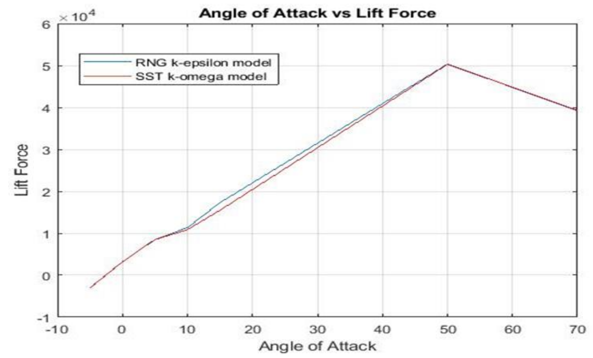

Fig. 8. Lift Force - Angle of Attack

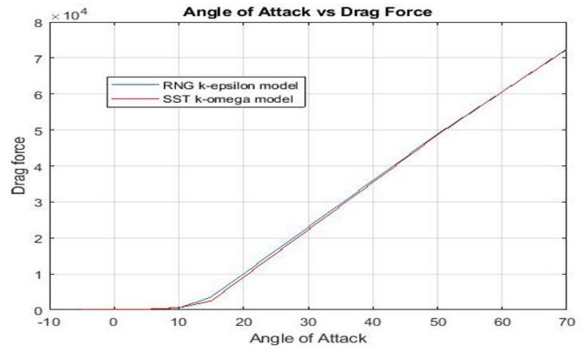

Fig. 9. Drag Force - Angle of Attack

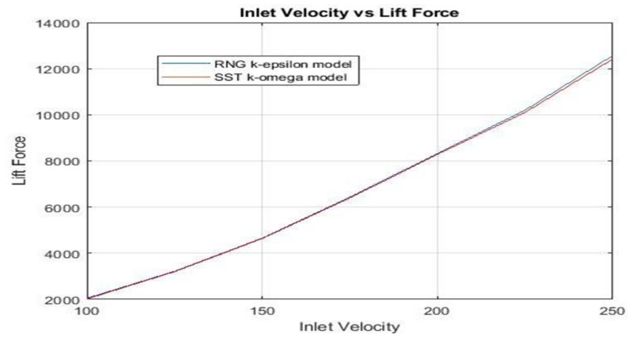

Fig. 10. Lift Force - Inlet Velocity

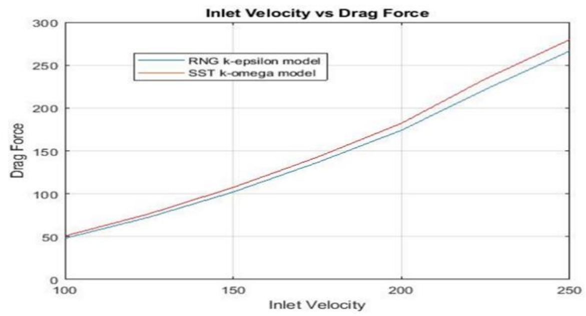

Fig. 11. Drag Force - Inlet Velocity

- For Boeing 737 aerofoil using RNG k-epsilon and SST k-omega model, the variations of lift and drag force with angle of attack and inlet velocity are represented using graphs.

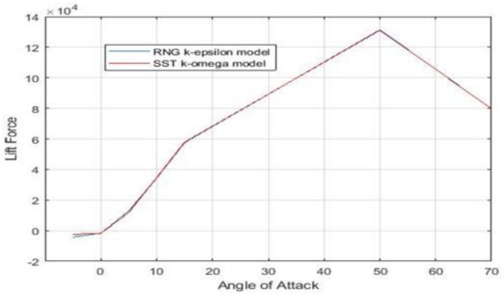

Fig. 12. Lift Force - Angle of Attack

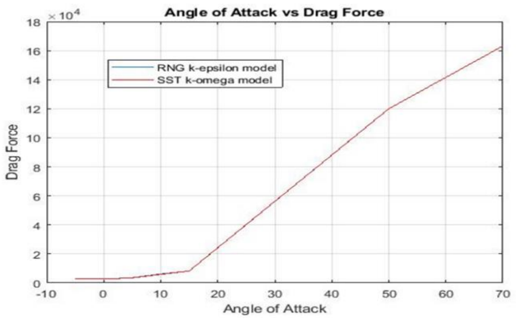

Fig. 13. Drag Force - Angle of Attack

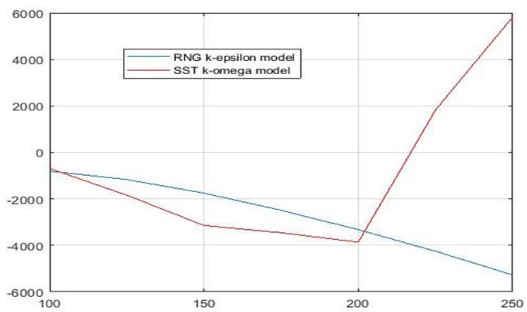

Fig. 14. Lift Force - Inlet Velocity

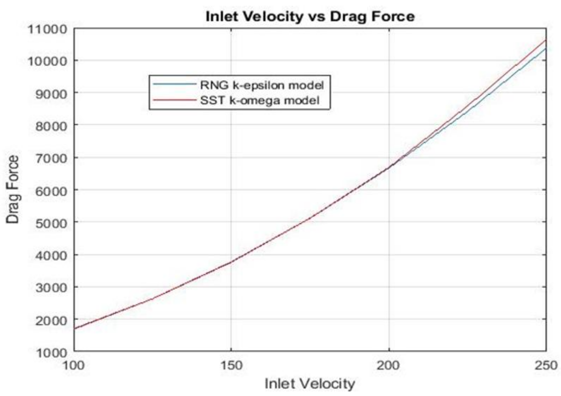

Fig. 15. Drag Force - Inlet Velocity

- It is observed that between NACA 2412 and Boeing 737, Boeing 737 aerofoil can be used where inlet velocity will be much higher generation of otherwise positive lift force is impossible with 0^0 degree of angle of attack.

- Boeing 737 generates higher lift force with the increment of angle of attack compared to NACA 2412 aerofoil.

- Based on the present result for a general instance (Angle of Attack = 5^0 and Inlet velocity =125 m/s), the lift to drag ratio for NACA 2412 is more than that of Boeing 737 and it is better suited for generating sufficient lift force at normal condition.

ACKNOWLEDGMENT

We are very much thankful to each other for contributing our best in this pandemic and whichever was possible has been done by us. This is an outcome of our research project relevant to the engineering applications. Our team work has inspired us to do research work farther on this particular field.

REFERENCES

[1] Mustafa Yılmaz, Hasan Koten, Erkan Çetinkaya, Ziya Coşar, A comparative CFD analysis of NACA0012 and NACA4412 airfoils, Journal of Energy Systems, ISSN:2602-2052, 2018
[2] Liyana Kharulaman, Abdul Aabid, Fharukh Ahmed Ghasi Mehaboobali, Sher Afghan Khan, Research on Flows for NACA 2412 Airfoil using Computational Fluid Dynamics Method, International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, October 2019
[3] Douvi C. Eleni, Tsavalos I. Athanasios, Margaris P. Dionissios, Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil, Journal of Mechanical Engineering Research, ISSN 2141-2383, March 2012
[4] Mohamed AboZeed Mohamed Atiah, Ahmed Wael Mansour Abd-Elkader, Mohamed Essam Ali Ashour, Boeing 737-700NG Airplane Flow Simulation, 4th IUGRC International Undergraduate Research Conference, July 2019
[5] MD. Safayet Hossain, Muhammad Ferdous Raiyan, Mohammed Nasir Uddin Akanda, Nahed Hassan Jony, A COMPARATIVE FLOW ANALYSIS OF NACA 6409 AND NACA 4412 AEROFOIL, International Journal of Research in Engineering and Technology, Oct-2014
[6] Mohamed A. Fouad Kandil, Abdelrady Okasha Elnady, Performance of GOE-387 Airfoil Using CFD, International Journal of Aerospace Sciences, 2017
[7] Mragank Pratap Singh, Dr. Mahendra Singh Khidiya, Sahil Soni, CFD Simulation of an Airfoil at different Angle of Attack, International Journal of Scientific Research in Science, Online ISSN : 2394-4099, 2016
[8] Ovais Gulzar, Saqib Gulzar, Sanjay Bhatele, Neelesh Soni, Impact of Variation in Angle of Attack on NACA 7420 Airfoil in Transonic Compressible Flow Using Spalart- Allmaras Turbulence Model, International Journal of Research in Mechanical Engineering & Technology, ISSN : 2249-5762 (Online), 2014