



e-ISSN: 2319-8753 | p-ISSN: 2320-6710

IJRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY

INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 9, Issue 9, September 2020

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 7.512

9940 572 462

6381 907 438

ijirset@gmail.com

www.ijirset.com

Computational Analysis of flow over NACA 6409 and NACA 8409 in Subsonic Region

Chaitanya Shinde¹, Sourabh Suryawanshi², Rushikesh Patil³

Former Student, Department of Mechanical Engineering, PVG's College of Engineering and Technology, Pune, Maharashtra, India^{1,2,3}

ABSTRACT: The analysis of three dimensional(3D) flow over two airfoils NACA 6409 and NACA 8409 is performed in subsonic region under the same flow conditions using ANSYS Fluent. The simulation procedure used for the purpose is validated with NASA Langley Research Centre validation case for NACA 0012. The Spalart Allmaras turbulence model is used to simulate the conditions and predict the flow around the airfoil. The geometry of airfoil is created in SolidWorks using airfoil coordinates. The computational domain comprised of 7500000 cells. Structured C-grid meshing of the flow field is done. The boundary conditions for simulation are mimicked from those used at NASA Langley Research Centre. The validated method is used for simulation of NACA 6409 and NACA 8409 airfoils at 0.5Mach and 0.9Mach speed. The flow at this speed becomes under compressible flow. Aerodynamic characteristics of both the airfoils like coefficient of lift(CL), coefficient of drag(CD), CL/CD ratio are studied at 0 angle of attack.

KEYWORDS: NACA 6409, NACA 8409, lift coefficient(CL), drag coefficient(CD), mach number, spalartallmaras turbulence model.

I. INTRODUCTION

An airfoil is a cross section of a wing or blade. An airfoil experiences aerodynamic forces when moving through a fluid. There are two types of airfoils: symmetrical airfoils and asymmetrical airfoils. Symmetrical airfoils have symmetric cross section about the chord line whereas asymmetrical airfoils have asymmetric cross section about the chord line. When an airfoil shaped body moves through a fluid, pressure variation is observed over the upper and lower surfaces. Due to this pressure differential aerodynamic forces like lift force and drag force are generated. These aerodynamic forces vary with angle of attack and free stream velocity. CFD is an economically viable and effective alternative to study aerodynamic characteristics of airfoils compared to experimental studies. In this study, the method adopted for simulation is validated with the help of published standard data by NASA for NACA 0012 airfoil. This method is used to predict the aerodynamic characteristics of NACA 6409 and NACA 8409 airfoils. Both the airfoils are cambered airfoils. Hence they generate lift force even at zero degree angle of attack.

Airfoil terminologies :

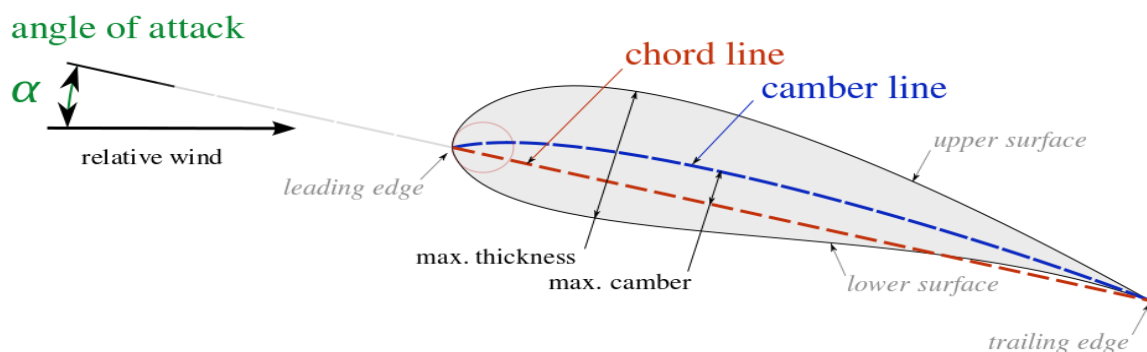


Fig.1 Airfoil terminologies

- **Leading edge** - The front edge of an airfoil
- **Trailing edge** - The rearmost edge of an airfoil
- **Chord line** - The straight line connecting the leading edge and trailing edge of an airfoil
- **Chord length** - The distance between the leading edge and trailing edge of an airfoil



- **Angle of attack** - The angle measured between the resultant relative wind and chord line of an airfoil
- **Suction Side** – The side with higher velocity and lower static pressure
- **Pressure Side** - The side with lower velocity and higher static pressure
- **Camber line** - The mean camber line is the locus of points midway between the upper and lower surfaces
- **Lift** - The component of aerodynamic force acting in the direction perpendicular to the motion of body moving through the fluid
- **Drag** - The component of aerodynamic force acting in the direction parallel to the motion of body moving through the fluid
- **Reynolds number** – It is a dimensionless number which indicates whether the flow is laminar or turbulent. It is the ratio of inertia force to viscous force.
- **Mach number** – Ratio of speed of body to the speed of sound in a medium

II. METHODOLOGY

The present study deals with compressible flow over NACA 6409 and NACA 8409 in subsonic region. The simulation is done using ANSYS Fluent. Since both the airfoils are cambered airfoils, they generate lift force even in case of zero angle of attack. In order to validate the procedure used for simulation, a standard profile NACA 0012 is simulated using the same procedure. The process for simulation involves 3 steps: Pre-processing, Solution and Post-processing. Pre-processing includes creation of geometry, defining the domain and discretization of the flow field into small cells called meshing. For geometry generation, coordinates of the airfoil profile are obtained from the standard equations from airfoil database [1]. 200 coordinates are used for each geometry to capture the curves properly. 100 for the upper profile and 100 for the lower profile. These coordinate files are imported in SolidWorks geometry modelling package to generate 2D contour. Contour is scaled to the chord length of 1m and extruded to the selected span length of 1m to create 3D model.

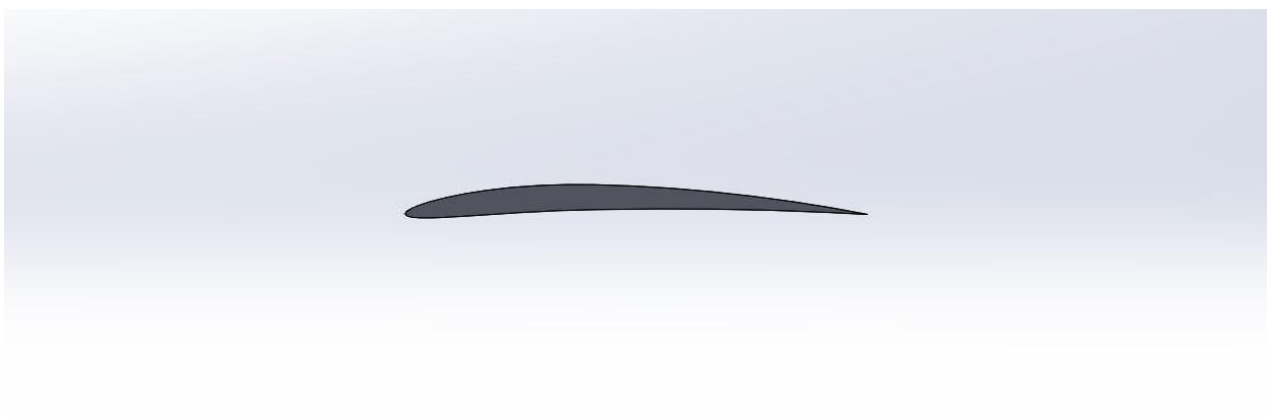


Fig.2 Geometry of NACA 6409

Mesh generation

For mesh generation, first domain is defined. Dimensions of domain are taken as 30 times the chord length on tail side of airfoil and 20 times of chord length in remaining all directions so that pressure far field condition can be applied. For 3D structured mesh generation ICEM CFD is used. In order to capture the boundary layer physics accurately, the mesh in vicinity of the boundary of the airfoil is made with small cells. First cell height is calculated from Y^+ calculations and 20 layers of inflation are created with growth rate of 1.2, so that boundary phenomenon responsible for lift and drag generation can be captured accurately. The mesh is then imported in ANSYS Fluent solver.

Sample first cell height calculation

Reynold number calculation

Mach no (M) = 0.5

Density of air (ρ) = 1.225 kg/m³

Dynamic viscosity (μ) = 1.8325 * 10⁻⁵

L = characteristic length

R_e = Reynold number

V_s = Velocity of sound

U^* = Friction velocity

$$V_s = \sqrt{\gamma * R * T}$$

$$= \sqrt{1.4 * 287 * 300}$$

$$= 347.1 \text{ m/s}$$

$$V = M * V_s$$

$$= 0.5 * 347.1$$

$$= 173.55 \text{ m/s}$$

$$R_e = \frac{\rho * V * L}{\mu}$$

$$R_e = 1.9722 * 10^6$$

- Calculation for First layer thickness

$$Y^+ = \frac{U^* * Y}{\nu}$$

$$U^* = \sqrt{\frac{T_w}{\rho}}$$

$$T_w = \frac{1}{2} * \rho * V^2 * C_f$$

$$= 61.95 \text{ N/mm}^2$$

$$U^* = \sqrt{\frac{61.95}{1.225}}$$

$$= 7.1116 \text{ m/s}$$

$$Y^+ = 1 = \frac{7.1116 * Y}{\frac{\mu}{\rho}}$$

$$Y = 2.1 * 10^{-6} \text{ m}$$

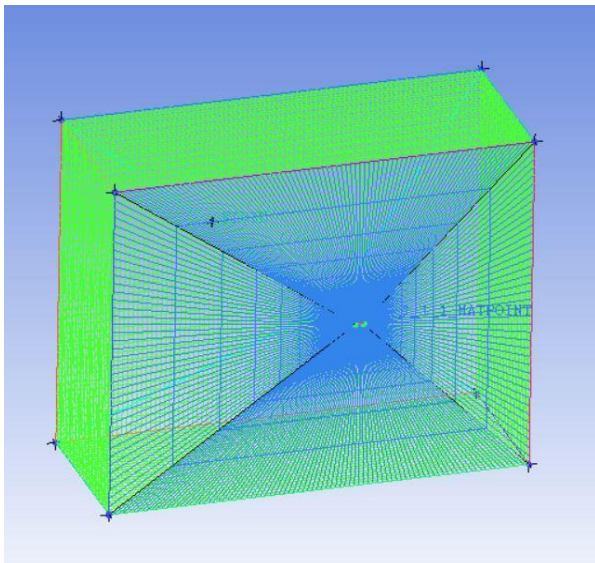


Fig.3 NACA 6409 mesh

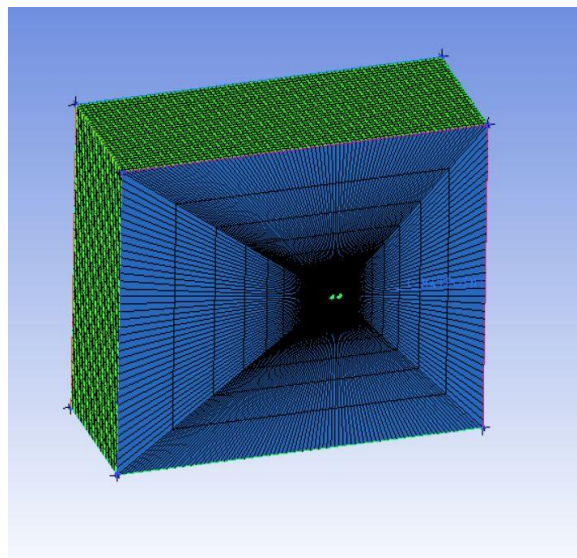


Fig.4 NACA 8409 mesh

Turbulence model

A turbulence model is required to predict the turbulent flow. The Spalart–Allmaras model is a one-equation model that solves a modelled transport equation for the kinematic eddy turbulent viscosity [2]. The Spalart–Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients [2]. The computation time required for solving external flow problems using Spalart–Allmaras is less compared to other turbulence models. Since the viscosity-affected region of the boundary layer is properly resolved ($y^+ \sim 1$ meshes), this model predicts the aerodynamic characteristics accurately.

Boundary conditions

The two main boundary conditions used for this simulation are Pressure far-field boundary condition and Wall boundary condition. Pressure far-field is applied to external domain. The velocity is 0.5 Mach with x component as 1 and y component as 0 for 0° angle of attack. For second case velocity of 0.9 Mach is used in this study. Gauge pressure is set to 0 pascal turbulent viscosity ratio is set at 10. Wall boundary condition is applied to the boundary of the airfoil.

Solver

For high Mach numbers ($M > 0.3$) or more specifically compressible flow calculations, density based solver is the apt choice. In the density-based approach, the continuity equation is used to obtain the density field while the pressure field is determined from the equation of state. The density-based solver solves the governing equations of continuity, momentum, and (where appropriate) energy and species transport simultaneously (i.e., coupled together) [3]. Governing equations for additional scalars will be solved afterward and sequentially. In the density-based solution methods, the discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow field [3].

III. VALIDATION OF SIMULATION PROCESS

Validation of simulation process is imperative to ensure that process yields fairly accurate results. NACA 0012 airfoil is utilized for validation since a lot of experimental data is available about it. The boundary conditions and operating conditions used are identical to the operating conditions of NASA Langley Research Centre validation cases [4]. Operating conditions for simulation are:

Parameter	Value
Reynold's number	6 million
Temperature	300K
Density	1.225 kg/m ³
Viscosity	1.7894x10 ⁻⁵ kg/ms

A segregated, implicit solver, ANSYS Fluent is used for the validation case. The 2D airfoil profile is generated in SolidWorks and boundary conditions, meshes are created in the ANSYS Fluent meshing module. Near the boundary of the airfoil, meshing is finer and dense in order to achieve superior computational accuracy.

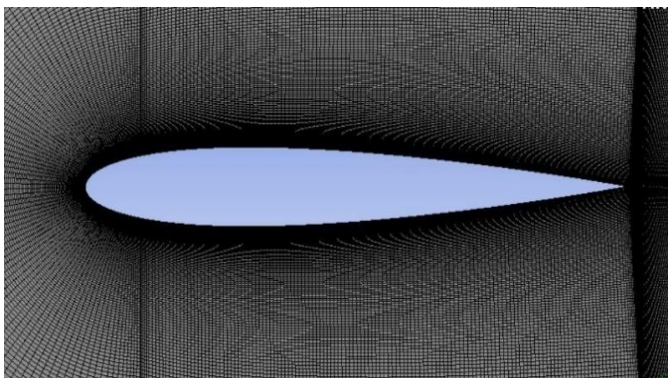


Fig.5 Structured mesh of NACA 0012

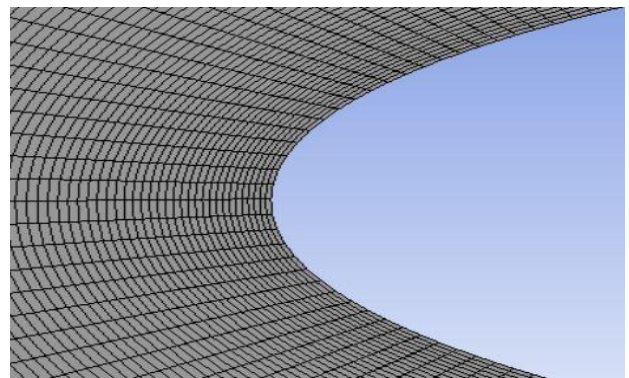


Fig.6 Detailed view of mesh

Structured mesh is created in ANSYS Fluent meshing module. Finer meshing is done in the region near the airfoil boundary. A C-grid mesh with 347673 elements is created to produce more accurate results. Spalart-Allmaras turbulence model is employed to predict the turbulent flow. A fully developed turbulent flow is simulated using ANSYS Fluent in accordance with the conditions used by NASA Langley Research Centre. Abbott and von Doenhoff data (Abbott, I. H. and von Doenhoff, A. E., "Theory of Wing Sections," Dover Publications, New York, 1959) was not tripped [6]. For fully developed turbulent flow, the Ladson tripped data (Ladson, C. L., "Effects of Independent Variation of Mach and Reynolds Numbers on the Low-Speed Aerodynamic Characteristics of the NACA 0012 Airfoil Section," NASA TM 4074, October 1988) comes out to be the most appropriate of these data sets for $Re = 6$ million.

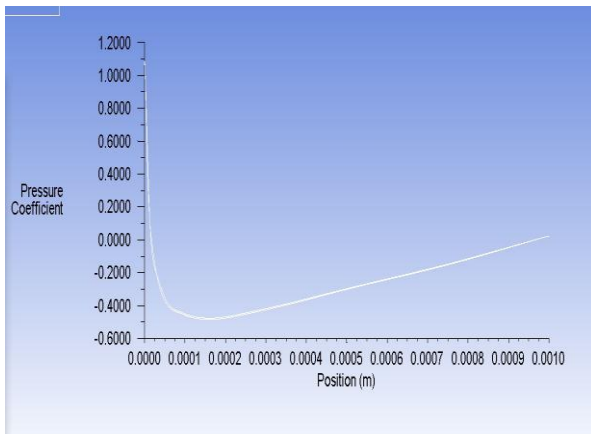


Fig.7 Cp vs X/C plot at 0 degree AoA

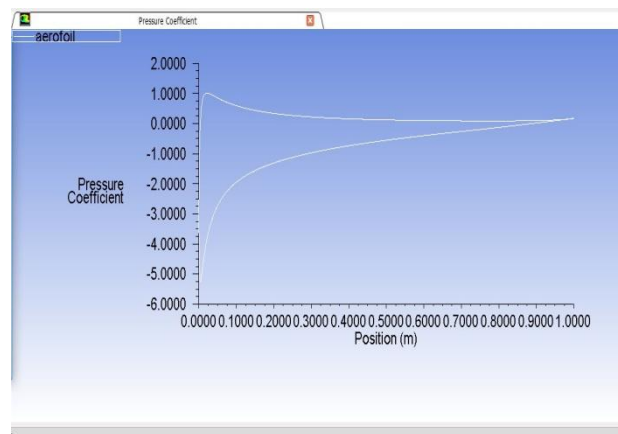


Fig.8 Cp vs X/C plot at 10 degree AoA

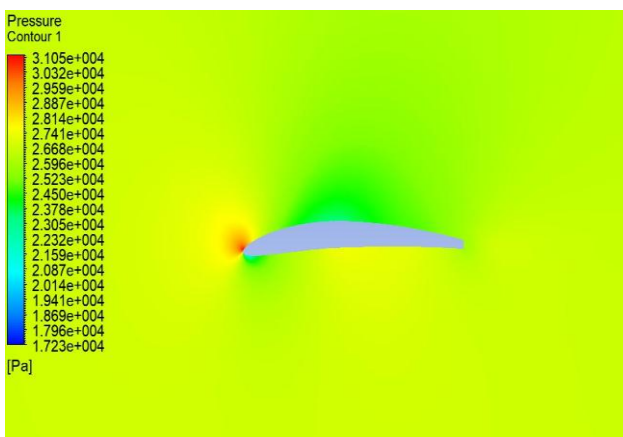
The solution converged after 559 iterations. The coefficient of pressure was plotted versus position on profile. The plot obtained from simulation of airfoil was overlapped on NASA’s data and the variation between corresponding points was observed to be less than 5%. This variation is due to perfect smooth surface of airfoil considered in simulation process. For 10 degree angle of attack, the simulation results closely conform to the experimental data Ladson[7]. For zero angle of attack, the simulation results match exactly with experimental study near the leading edge and trailing edge but there is slight deviation (<5%) at the middle section of airfoil which is acceptable.

It reveals that results obtained by methodology adopted for simulation are closely matching with the standard data, thus the procedure is validated.

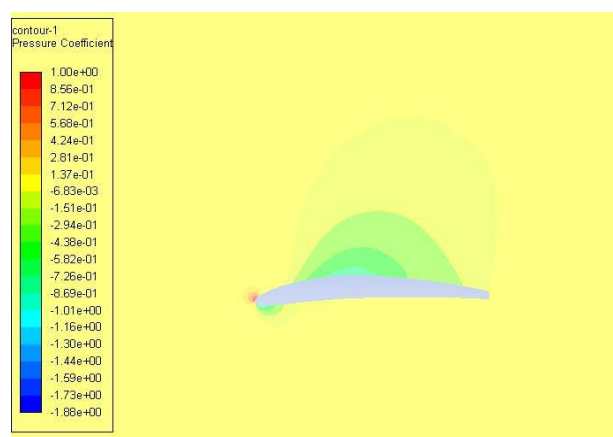
IV. RESULTS

NACA 6409 and NACA 8409 airfoils are simulated for the flow velocity of 0.5Mach and 0.9Mach using Spalart-Allmaras turbulence model by the method validated above.. In order to avoid manufacturing issues, the trailing edge is cut in last 5% of chord. Since the flow velocity involved is greater than Mach 0.3, the flow is compressible. The density-based solver predicts the flow parameters accurately and helps in reducing the computational effort. The results of simulation include Pressure contours and velocity contours as obtained using ANSYS Fluent. The aerodynamic characteristics of prime importance in this study are lift coefficient (CL), drag coefficient (CD) and CL/CD ratio. These characteristics for each case are obtained and analysed. The results of simulation for 0.5 Mach number are attached.

i. NACA 6409

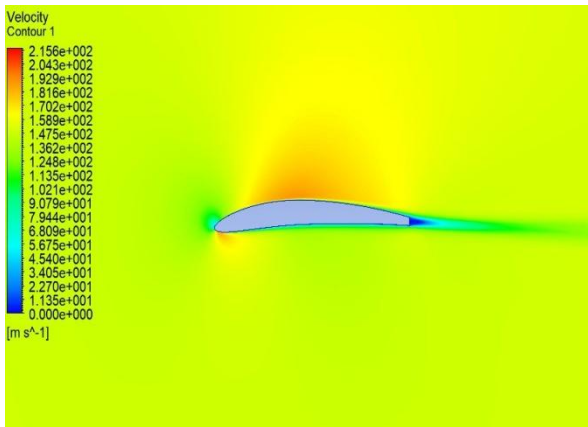


(a) Pressure contour

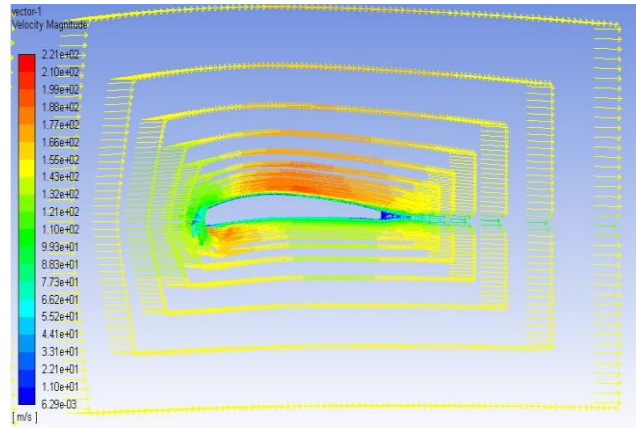


(b) Pressure coefficient

Pressure is maximum at the leading edge due to stagnation of flow at leading edge. Pressure decreases on the upper side from the leading edge to middle of airfoil. After this there is slight increase in pressure from middle to trailing edge. While on the lower side pressure keeps on increasing from leading edge to trailing edge. Because of this pressure difference between upper and lower surfaces lift force is generated.



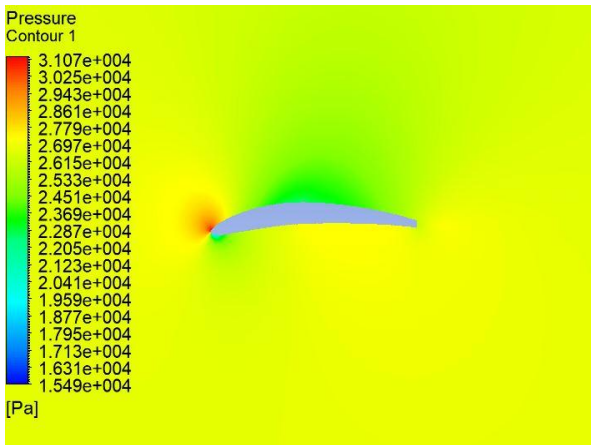
(a) Velocity contour



(b) Velocity vectors

Velocity of flow is zero at the leading edge at the same point where pressure is maximum. Velocity over the upper surface is higher due to lesser pressure. Velocity increases till the middle of airfoil and thereafter starts decreasing gradually. On the lower surface, velocity is low as compared to upper surface.

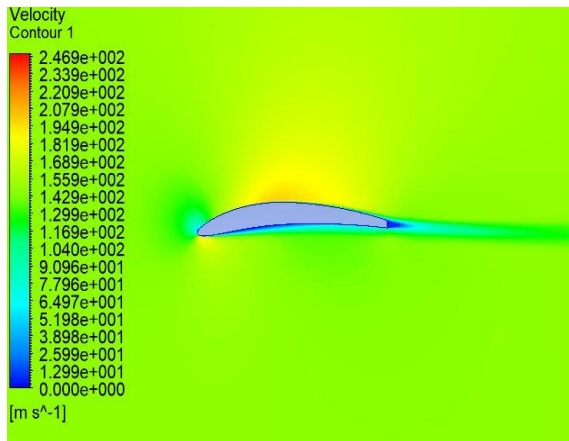
ii. NACA 8409 contours



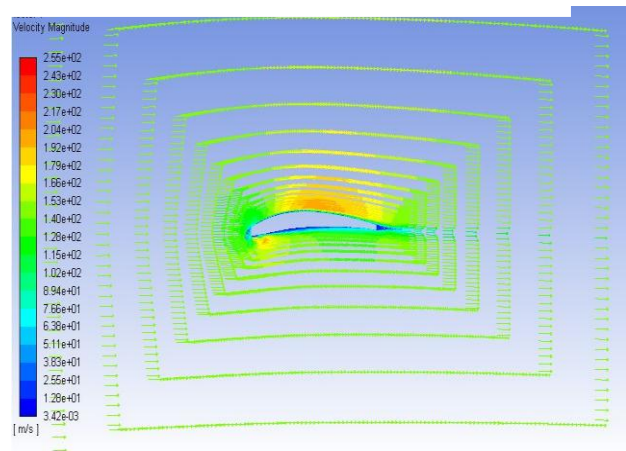
(b) Pressure contour



(b) Pressure coefficient



(b) Velocity contour



(b) Velocity vectors

V.CONCLUSION

The study included 3D simulations of two NACA profiles viz. NACA 6409 & NACA 8409 at 0 degree angle of attack and velocity of Mach 0.5 and Mach 0.9 in the sub-sonic region. The results are successfully computed for lift generated by the airfoil and the contours of pressure and velocity magnitude have been plotted. Contours of static pressure, Mach number, pressure coefficient, pressure variation over airfoil as well as contour of velocity magnitude have been studied and analyzed. They are found coherent with the expected phenomenon.

NACA 6409 and NACA 8409 both are cambered airfoils and hence generate lift force even at zero degree angle of attack. The airfoils deflect the flowing fluid and as a reaction, the fluid exerts a force on the airfoil which is the aerodynamic force. This force when resolved into vertical and horizontal components gives the lift and drag force. It is evident from the results that performance of both NACA 6409 and NACA 8409 is similar at velocity of Mach 0.5. The lift coefficient of NACA 8409 is higher compared to NACA 6409. However, the CL/CD ratio for both airfoils is very close at this velocity as the drag coefficient for NACA 8409 is higher than that of NACA 6409. This variation in the aerodynamic characteristics is due to difference in cambers of the airfoils under study. Maximum camber for NACA 8409 is 8% and that for NACA 6409 is 6% of chord.

The results show that when the velocity is increased to Mach 0.9, the lift coefficient decreases significantly. A slight increase is observed in the drag coefficient which results in decrease in the overall CL/CD ratio at this velocity. Although the drag coefficients of both the airfoils at this velocity are same, the lift to drag ratio of NACA 6409 is higher in comparison to NACA 8409. NACA 6409 has better aerodynamic characteristics at Mach 0.9 velocity. Thus it can be concluded that the aerodynamic performance of both the airfoils is similar in the subsonic region. With higher Mach numbers NACA 6409 has clear aerodynamic advantage over NACA 8409.

REFERENCES

- [1] Airfoil database at : <http://www.airfoiltools.com>
- [2] Information on Spalart-Allmaras One equation turbulence model at : https://en.wikipedia.org/wiki/SpalartAllmaras_turbulence_model
- [3] ANSYS Fluent 12.0 Theory guide
- [4] 2D NACA 0012 Airfoil Validation Cases available at: http://turbmodels.larc.nasa.gov/naca0012_val.html.
- [5] John. D Anderson, "Fundamentals of Aerodynamics", Fifth Edition, McGraw Hill Inc.
- [6] Abbott, I. H. and von Doenhoff, A. E., "Theory of Wing Sections," Dover Publications, New York, 1959.
- [7] Ladson, C. L., Hill, A. S., and Johnson, Jr., W.G., "Pressure Distributions from High Reynolds Number Transonic Tests of an NACA 0012 Airfoil in the Langley 0.3-Meter Transonic Cryogenic Tunnel,"



INNO SPACE
SJIF Scientific Journal Impact Factor

**Impact Factor:
7.512**

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 9940 572 462  6381 907 438  ijirset@gmail.com



www.ijirset.com

Scan to save the contact details