Prediction of Pressure, Velocity Distributions and Visualization of Flow Patterns Around NACA-2415 Aerofoil for Various Angles of Attack

Vipinkumar c p

M.Tech, Thermal and Fluids Engineering
Department of Mechanical Engineering
L.B.S College of Engineering, Kasaragod, Kerala, India

Abstract

Determining the aerodynamic characteristics of aerofoil profiles is an important practical problem of Computational aerodynamics. The aim of the proposed work is to predict pressure and velocity distributions around NACA-2415 aerofoil and visualize the patterns for different angle of attack for Mach number less than 0.3, which is incompressible flow. The aerofoil profile coordinates data available in the online library (Airfoil Tools website) and imported these coordinates into ANSYS design modular to create aerofoil geometry. ANSYS Fluent meshing will be use for generation of the computational meshes and Openfoam v6 used for the stimulating the case.

1. Introduction

An aerofoil is the cross sectional shape of wing. It produces an aerodynamic force while moving through a fluid. Aerofoil design plays a very important role in the designing of wings as many factors are dependent on it like the shape of the aerofoil decides the amount of lift that will be generated by it and the drag force and many other things. There are many types of aerofoil designs available like Clark Y, NACA, NASA GA (W) etc out of which the NACA aerofoils are the most standard aerofoils used in preliminary design and which are modified later for their specific use. There is extensive experimental data on many aerofoil shapes that has been done by the National Advisory Committee for Aeronautics (NACA).
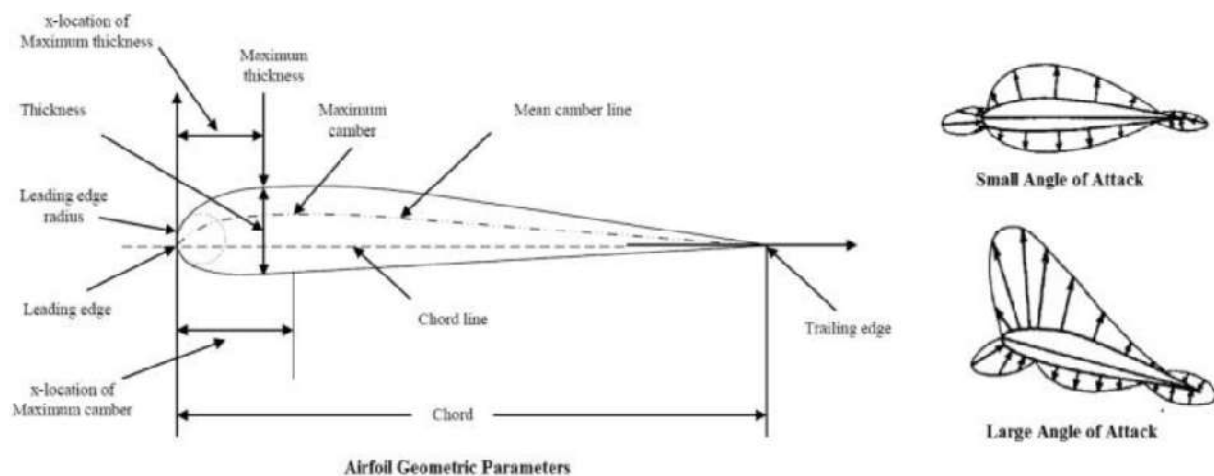

Figure 1: Aerofoil Geometric Parameters and Pressure variation with small and large angle of attack[1].

Geometric Parameters of an Aerofoil: It is a positive cambered section having thicker part at its front. An aerofoil-shaped body moved through the air will vary the static pressure on the top surface and on the bottom surface of the aerofoil. A typical aerofoil section is shown in (Fig. 1.), where several geometric parameters are illustrated. If the mean camber line in a straight line, the aerofoil is referred to as symmetric aerofoil, otherwise it is called cambered aerofoil. The camber of aerofoil is usually positive. In a positive cambered aerofoil, the upper surface static pressure is

less than the ambient pressure, while the lower surface static pressure is higher than the ambient pressure. This is due to the higher air speed at upper surface and lower speed at lower surface of the aerofoil. This pressure difference between upper and lower surfaces increases, as the angle of attack increases. The force divided by the area is called pressure, so the aerodynamic force generated by an aerofoil in a flow field may be calculated by multiplication of total pressure by area. The total pressure is simply determined by integration of pressure over the entire surface[1].

The National Advisory Committee for Aeronautics (NACA) developed a number of aerofoils. These aerofoils are described using a series of digits following the word "NACA". NACA 4-Digit Code:

a. First digit - maximum camber of the aerofoil as percentage of the chord.
b. Second digit - distance of maximum camber from the aerofoil leading edge in tens of percents of the chord.
c.Third and Fourth digit - maximum thickness of the aerofoil as percent of the chord .

The aim of this stimulation is to predict pressure and velocity distributions around NACA-2415 aerofoil and visualize the patterns for different angle of attack for Mach number less than 0.3, which is incompressible flow.

2. Problem Statement

The aim of the proposed work is to predict pressure and velocity distributions around NACA-2415 aerofoil and visualize the patterns for different angle of attack for Mach number less than 0.3, which is incompressible flow. The aerofoil profile coordinates data available in the online library (Airfoil Tools website) and imported these coordinates into ANSYS design modular to create aerofoil geometry. ANSYS Fluent meshing will be use for generation of the computational meshes and Openfoam v6 used for the stimulating the case.Simple foam (Steady-state solver for incompressible, turbulent flow, using the SIMPLE algorithm) will be used as the solver .The geometry and Flow domain shown in figure 2.
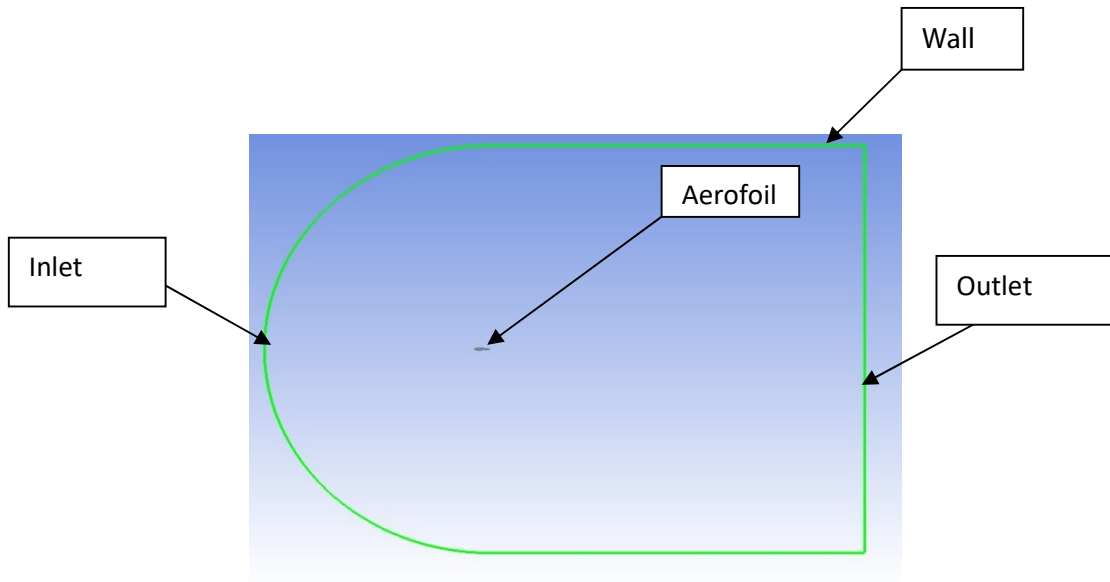
Figure 2. The geometry and Flow domain

The boundary conditions for the problem has stated below :

- Inlet velocity U = 27 m/s

- Angle of attack: 4° ,8° ,12°, 16°, 20°

- RAS Model: SpalartAllmaras

- Kinematic viscosity: $1.714*10^{-5}\ m^2/s$

- Transport Model: Newtonian

- Solver: simpleFoam

3. Governing Equations

The simpleFoam solver employs the SIMPLE algorithm

Continuity equation: $\nabla \cdot u = 0$

Momentum equation: $\nabla \cdot (u \otimes u) - \nabla \cdot R = -\nabla p + Su$

Where, u -velocity; p-kinematic pressure; R=stress tensor, Su=Momentum source

Turbulence Model

The Spalart–Allmaras model is mainly used in aerospace applications involving high curvature, and adverse pressure gradient. It is a one-equation model that solves a modelled transport equation for the kinematic eddy viscosity.

The one-equation model is given by the following equation:

$$\frac{\partial \hat{v}}{\partial t} + u_j \frac{\partial \hat{v}}{\partial x_j} = c_{b1}(1 - f_{t2})\hat{S}\hat{v} - \left[c_{\omega 1}f_\omega - \frac{c_{b1}}{k^2}f_{t2}\right]\left(\frac{\hat{v}}{d}\right)^2 + \frac{1}{\sigma}\left[\frac{\partial}{\partial x_j}\left((v + \hat{v})\frac{\partial \hat{v}}{\partial x_j}\right) + c_{b2}\frac{\partial \hat{v}}{\partial x_j}\frac{\partial \hat{v}}{\partial x_i}\right]$$

and the turbulent eddy viscosity is computed from:

$$\mu_t = \rho\hat{v}f_{v1}$$

where

$$f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3}$$

$$\chi = \frac{\hat{v}}{v}$$

and ρ is the density, μ is the molecular kinematic viscosity,

4. Stimulation procedure

4.1 Geometry and Mesh

Geometry creation

The geometry used in this study is the NACA 2415 aerofoil. The main objective is to find out the pressure and velocity pattern distribution over the aerofoil. The geometrical data points for NACA 2415 aerofoil is downloaded from aerofoiltools.com .The data points then cleaned and imported to Ansys design modular for geometry creation. The geometry is created using 3D curve feature in the Ansys Design Modular.

Computational Domain

For studying the aerodynamics characteristics of the aerofoil, different types of computational domains are used, C and O domains being one among the most popular ones. For the present 4study, C domains are used for computation. This task is done by modeling a steady flow passing a stationary aerofoil.The computational domain selected is 15 times chord length to inlet 15 times length to top and bottom walls and 25 times chord length to outlet. The C-type structured grid with 49095 quad elements is generated using Ansys Fluent Meshing . The enhanced wall treatment approach is used as the near-wall treatment method to resolve the near-wall region including the viscous sub layer. The boundary conditions which are used for the computational domain boundaries are inlet velocity boundary condition at the inlet and the sides of the domain ,walls of the aerofoil and outlet boundary condition with atmospheric pressure at the domain exit. The computational domain and grid arrangement which are used in the current study is shown in figure 3 and figure 4.
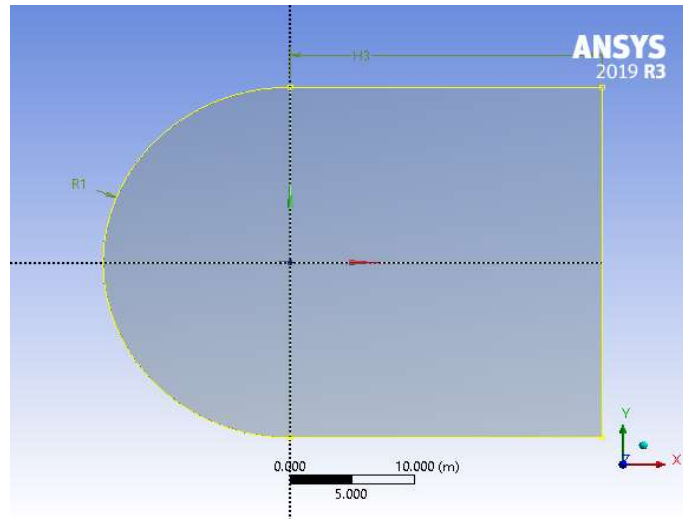
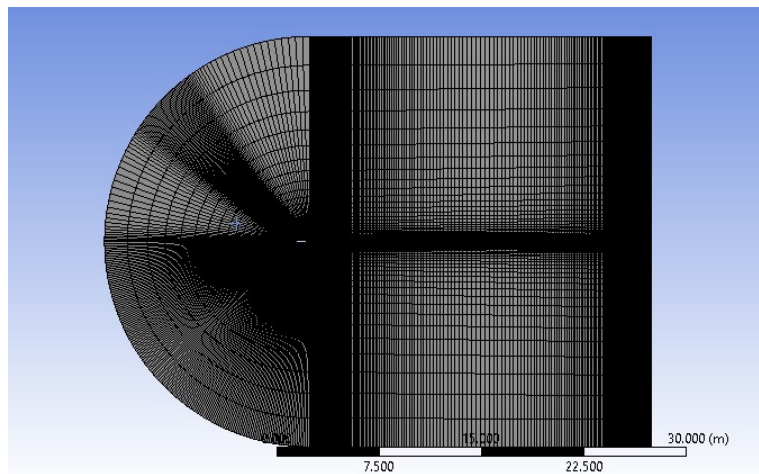Figure 3**:** A sketch of C computational domain used to study flow around an aerofoil

Figure 4 : Meshed sketch of C computational domain used to study flow around an aerofoil

The geometry and mesh were built in Ansys academic and converted into openfoam mesh using openfoam utilities listed below

fluentMeshToFoam

fluentMeshToFoam is used to Convert a Fluent mesh (in ASCII format) to foam format including multiple region and region boundary. It can be run by a command fluentMeshToFoam Synopsis : fluentMeshToFoam FLUENT_MESH , Convert a mesh file FLUENT_MESH from Fluent format to foam format. fluentMeshToFoam can only handle the following 2D cell types: tri, quad and 3D cell types: tet, hex, pyramid, prism.

checkMesh

checkMesh is a utility which analyzes and evaluates mesh statistics and quality parameters. It can be run anytime by a command checkMesh > log.checkMesh .The log contains information Statistics about number of points, faces, cells, etc.

4.2 Initial and Boundary Conditions

The variables of particular interest here are the velocity, pressure, and the parameters required for the Spalart-Allmaras Turbulence Model, density, kinematic viscosity .The velocity to be given at the inlet is given value 27 m/s and remaining velocity for different angle determined below
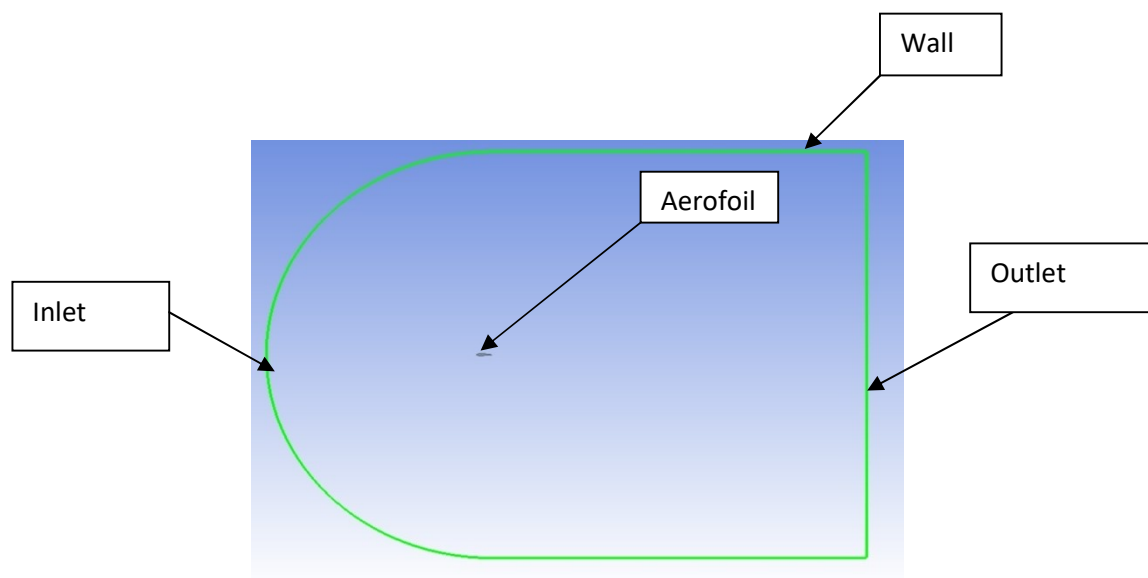

Figure 5: Boundary conditions for domain

4.2.1 Transport properties

Kinematic viscosity for the fluid(air) is assumed as $1.714 \ 10^{-5} \frac{m^2}{s}$,density as $1.225 \ \frac{kg}{m^3}$ and the Reynolds number for the flow is described as $1.5 * 10^6$.

4.2.2 Velocity conditions

The velocity conditions for aerofoil computational domain are described below.

Boundary	Inlet Conditions
inlet	freestreamVelocity
outlet	freestreamVelocity
walls	noSlip
aerofoil	noSlip
Wall surface_body	noSlip

freestreamVelocity velocity conditions

Conditions for freestreamVelocity velocity at different angle of attack calculated as follows

$$tan\theta = (\frac{u_y}{u_x})$$
$$u_y = u_x tan\theta$$

Angle of attack	$u_x(\frac{m}{s})$	$u_y(\frac{m}{s})$
$0°$	27	0
$4°$	26.934	1.883
$8°$	26.737	3.757
$12°$	26.409	5.613
$16°$	26.409	7.572
$20°$	26.409	9.612

4.2.3 Pressure conditions

The pressure conditions for aerofoil computational domain are described below.

Boundary	Inlet conditions
inlet	freestreamPressure
outlet	freestreamPressure
walls	zeroGradient
aerofoil	zeroGradient
Wall surface_body	zeroGradient

Wall functions are necessary to treat the flow near walls while turbulence models treat the flow far from walls. nutUSpaldingWallFunction used as wall function .This boundary condition provides a turbulent kinematic viscosity condition when using wall functions for rough walls .a value of 0.14 assigned to it.

4.2.4 Time and data input/output control

The OpenFOAM solvers begin all runs by setting up a database. The database controls Input/Output and, since output of data is usually requested at intervals of time during the run, time is an inextricable part of the database. The controlDict dictionary sets input parameters essential for the creation of the database. The keyword entries in controlDict are listed in the following sections. Only the time control and write Interval entries are mandatory, with the database using default values for any of the optional entries that are omitted. Entries from a controlDict dictionary are given below.

application	simpleFoam;
startFrom	startTime
startTime	0
stopAt	endTime
endTime	1000
deltaT	1
writeControl	timeStep
writeInterval	50

4.3 Solver

OpenFOAM is an open source commercial CFD code written in C + +. It was chosen to perform simulation of the conventional NACA2415 Aerofoil . It consists of various embedded libraries which are accessible for review and modifications. The libraries consists of numerous mathematical models and CFD tools organized in directories.

SimpleFoam Solver Specifications

The Navier-Stokes equations are analytical equations, hence it cannot be understood by the solver. In order to make the solver understand the equation, it has to be transferred in a discretized form. This process is known as discretization. The typical discretization methods are finite difference, finite element and finite volume methods. OpenFOAM is based on the Finite Volume Method approach. Simple foam (Steady-state solver for incompressible, turbulent flow, using the SIMPLE algorithm) will be used as the solver for NACA2415 problem .

Overview of simpleFoam solver

- Category: Incompressible
- steady state
- incompressible
- Turbulence
- Finite volume options

Input requirements for SimpleFoam solver:

Mandatory fields:

- p: kinematic pressure $[\frac{m^2}{s^2}]$
- U: velocity $[\frac{m}{s}]$

Physical models:

- Turbulence constant/turbulence Properties
- finite volume options constant/fv Options (optional)

5. Results and Discussions

Once stimulation has been done, the next task is to perform computation on the model that we have just created. In order to visualize and analyze the results various tools and softwares are required. In this stimulation Paraview used as a tool to visualization of result .ParaView is an open-source, multi-platform data analysis and visualization application. It can quickly build visualizations to analyze their data using qualitative and quantitative techniques. The data exploration can be done interactively in 3D or programmatically using ParaView's batch processing capabilities. The pressure and velocity distribution around the aerofoil was analyzed using Parafoam.

The velocity and pressure gradient plots for different angle of attacks ie $0°, 4°, 8°, 12°, 16°, 20°$ are shown in figures below .In all the cases the free stream velocity is taken to be 27 m/s , the fluid is taken as air with density 1.225 $\frac{kg}{m^3}$ and kinematic viscosity is taken as 1.714 $10^{-5} \frac{m^2}{s}$ .The aerofoil used is NACA 2415.The computational domain selected is 15 times chord length to inlet 15 times length to top and bottom walls and 25 times chord length to outlet. The noslip boundary conditions are given for aerofoil surface wall and free stream pressure condition for overall domain.
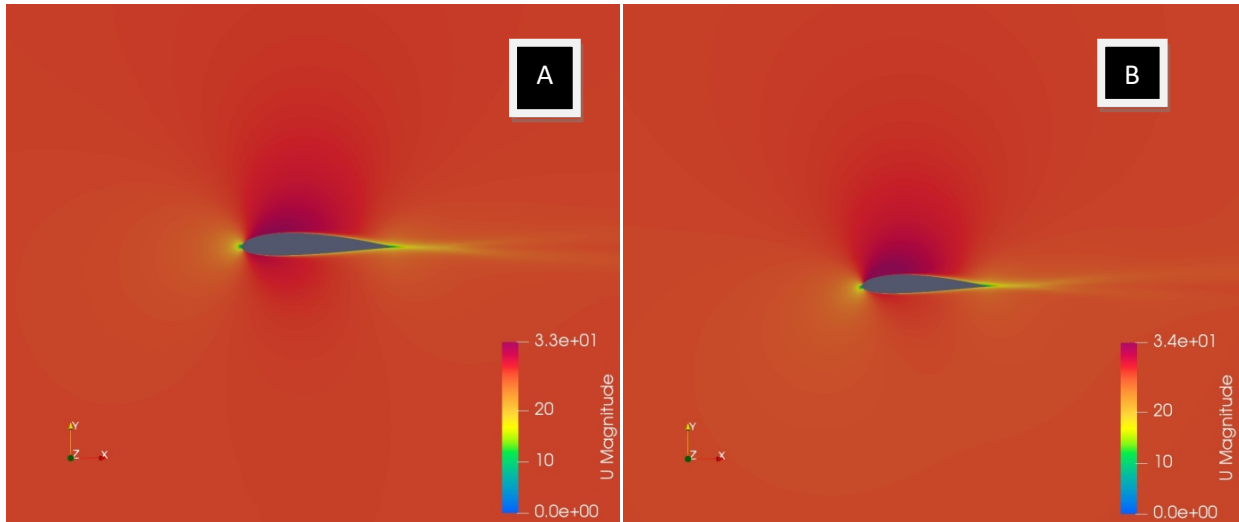
Velocity distributions plots

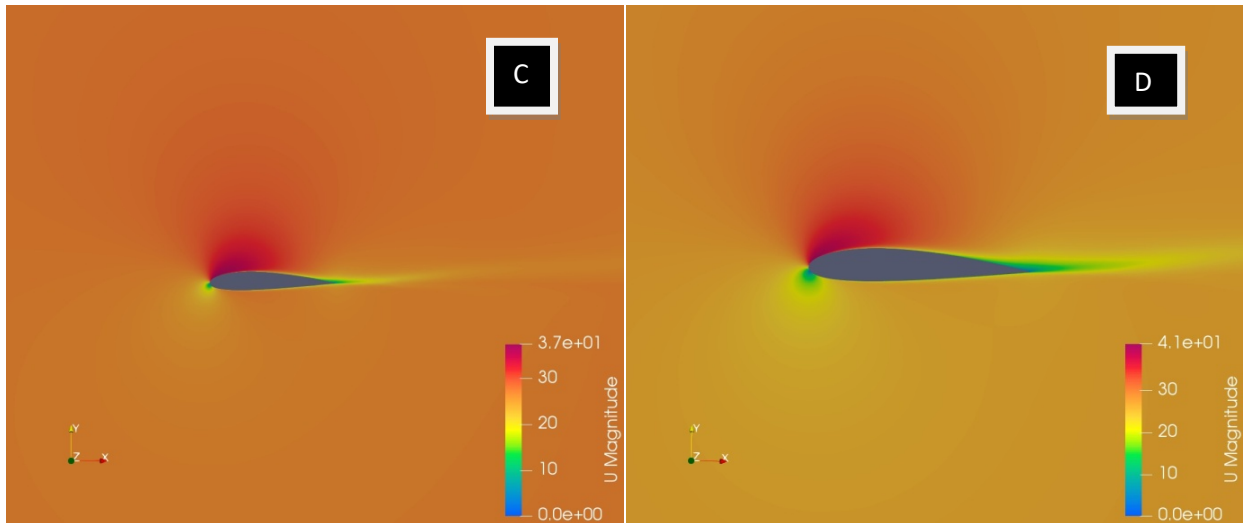Figure 6 : A, B Velocity distributions at angle of attacks $\alpha = 0°, 4°$

Figure 7 : C, D Velocity distributions at angle of attacks $\alpha = 8°, 12°$
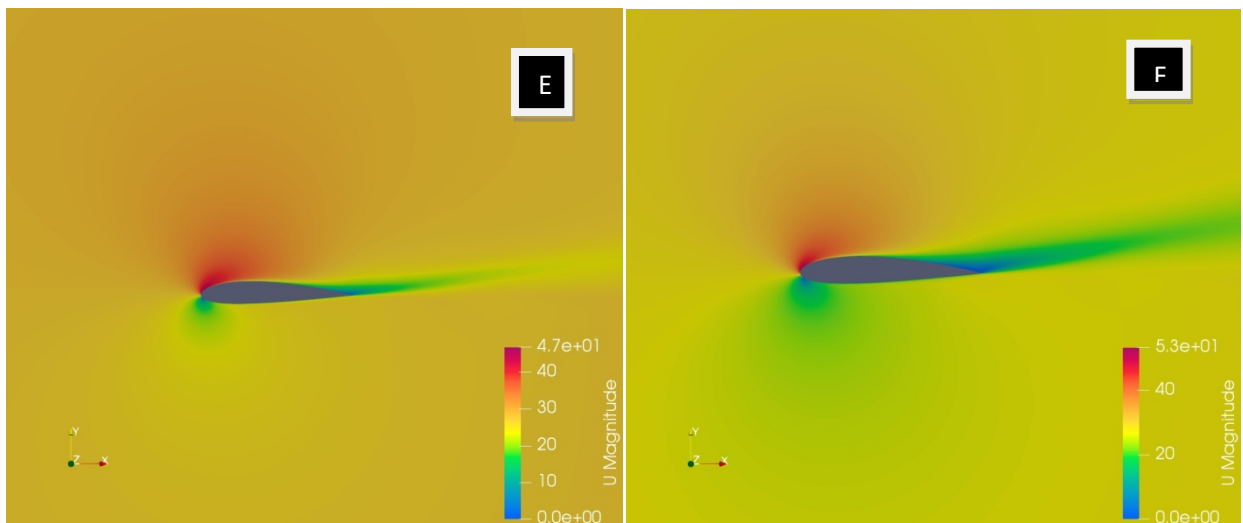

Figure 8 : E,F – Velocity distributions at angle of attacks $\alpha = 16°, 20°$

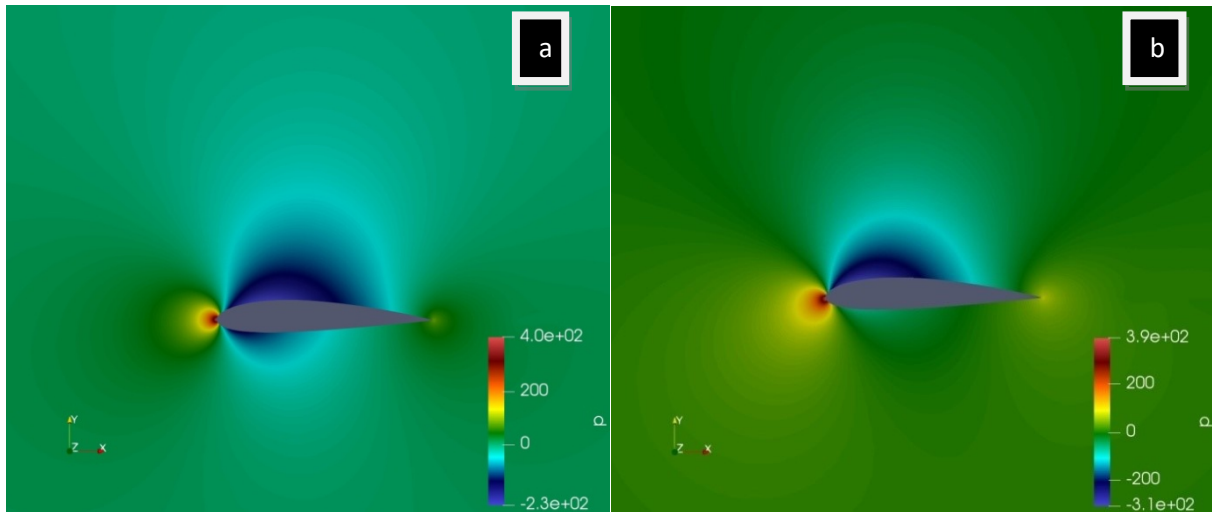**Pressure distributions plots**

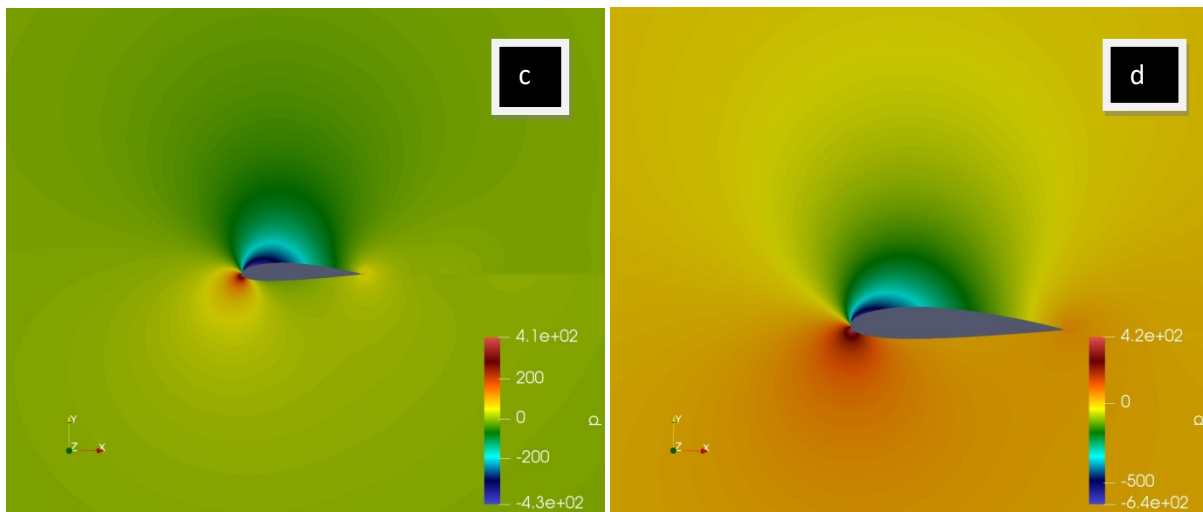Figure 9 : a, b - Pressure distributions at angle of attacks α = 0°, 4

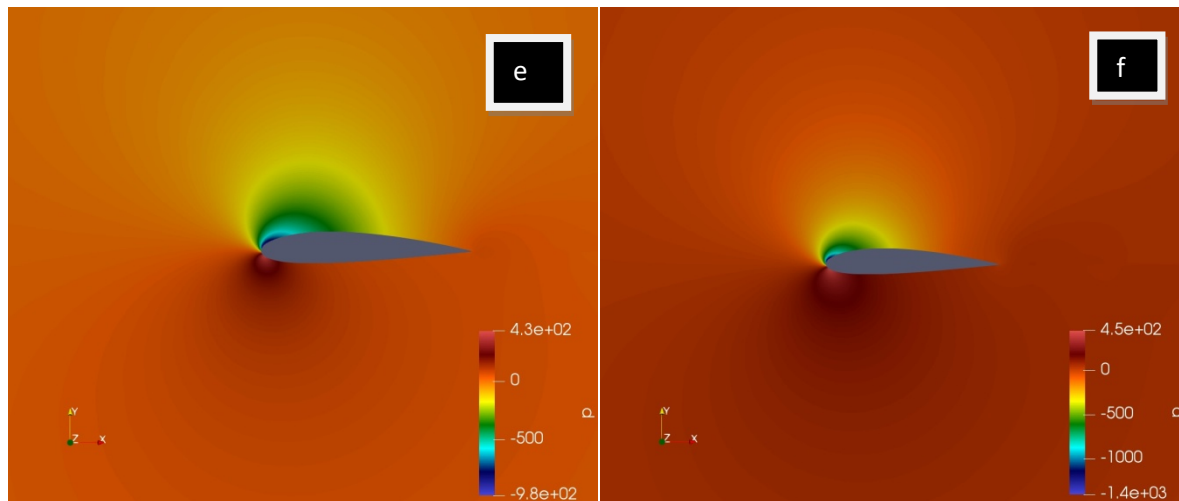Figure 10: c, d - Pressure distributions at angle of attacks α = 8°, 12

Figure 11: e,f - Pressure distributions at angle of attacks α = *16°, 20°*

In general these plots illustrate the development of the boundary layer as the pressure slowly increases after peak suction from the leading-edge to the trailing-edge. In order to show the evolution of the pressure and velocity as the angle of attack were increased the contours of pressure and velocity distribution over the NACA2415 aerofoil is depicted .At angle of attacks α = 0°, 4° the flow follows smoothly aerofoil surfaces with indication of pressure build up with leading edge(stagnation point). As the angle of attack increases it is visible that a low pressure region is developed on the top surface of the aerofoil which creates the lift .As angle of attack increased further that is greater than 8° there is wake region formation which separates flow further creates stall.

The aerodynamic performance of the aerofoil greatly depends on the angle of attacks. At higher angle of attacks the aerodynamic characteristics of the aerofoil varies marginally with the angle of attack. However, for low angle of attack, the aerodynamic performance of the aerofoil varies rapidly as the configuration and/or angle of attack changes. The aerofoil performance is relatively poor at low and high angle of attack number as compared to angle of attack at 8^0. The poor performance of aerofoil at high angle of attack is mainly because of the flow separation (boundary layer separation) which results in reduced lift and increased pressure drag on the aerofoil.

References

[1]. "Flow Simulation and Theoretical Investigation on Aerodynamics of NACA-2415 Aerofoil at Low Reynolds Number" SAE Technical Paper 2015-01-2576, 2015   Kumar, V., Tomar, V., Kumar, N., and Jain, S.

[2]. https://turbmodels.larc.nasa.gov/spalart.html

[3]. https://www.openfoam.com/documentation/guides/latest/doc/guide-turbulence-ras-spalart-allmaras.html

[4]. https://openfoamwiki.net/index.php/SimpleFoam

[5]. "Numerical investigation of fluid flow around NACA0012 airfoil using OpenFOAM", Subham Kant Das (https://cfd.fossee.in/case-study-project/completed-case-studies)