Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2011
(ICMERE2011) 22- 24 December 2011, Chittagong, Bangladesh

ICMERE2011-PI-163

NUMERICAL SIMULATION AND AERODYNAMIC PERFORMANCE COMPARISON BETWEEN TWO AIRFOILS

[1]Abdul Mannan, [2]Mr. A. S. M Sayem [3]Saemul Seraj, [4]Md. Elias Mollah,
[1]Student, Department of Mechanical Engineering, CUET, Bangladesh
[2]Assistant Professor, Department of Mechanical Engineering, CUET, Bangladesh
[3]Student, Department of Mechanical Engineering, CUET, Bangladesh
[4]Student, Department of Mechanical Engineering, CUET, Bangladesh

[1]mannan.cuetme06@gmail.com,[2]yessayem@yahoo.com,[3]jeorge2005@yahoo.com, [4]hellow_elias96@yahoo.com
Corresponding Author: [1]mannan.cuetme06@gmail.com

Abstract- *Numerical simulation of flow around airfoils is very much important for aerodynamic design of aircraft wings and turbomachinery components because the cost, time and difficulties of experiments can be reduced through numerical simulation. The precise shape of airfoil largely carries the performance of an aircraft and turbomachinery Key parameters that determine the performance of an airfoil are Pressure distribution, lift coefficients and drag coefficients which are computed at various angles of attack using FLUENT software at low Reynolds number. The fluid flow is analyzed over NACA 4412 airfoil and also compared with a redesigned airfoil. Pressure, lift and drag coefficients are highly influenced by the angle of attack. With increasing angle of attack the lift coefficient of redesigned airfoil increases but on the other hand the lift coefficient to drag coefficient ratio decreases. For both airfoils stalling occurs at* 15^0*angle of attack and the lift/drag ratio increases very rapidly up to about* 2^0 *or* 3^0*. After* 2^0 *or* 3^0*The drag coefficient increase more rapidly than lift coefficient so lift/drag ratio gradually decreases.*

Keywords: 4412 *Airfoil, lift coefficient, drag coefficient, boundary layer thickness, CFD, FLUENT, GAMBIT.*

INTRODUCTION

Precise shape of airfoil (the cross section of wing) is one of the most important considerations of optimum designs of aircraft. The precise shape of airfoil largely carries the performance of an aircraft. Best performance of aircraft means how smoothly it runs, people feels how much comfort; journey by aircraft is how much safe, etc. performance of airfoil depends on lift coefficient, drag coefficient, angle of attack, etc. For better performance, the drag coefficients should be less and the lift coefficients should be large.

An experimental investigation may be very time consuming, dangerous, prohibitively expensive, or impossible for another reason [1]. Computational study of fluid dynamics can overcome these drawbacks because it builds a 'virtual prototype' of the system or device that we want to analyze. The software will provide us images and data, which predict the performance of that design. The airfoil performance varies with changing shape of airfoil so; how this change in shape affects the performance of airfoil is analyzed in this project. Here fluid flow is analyzed for different angles of attack over 4412 airfoil and it is compared with redesigned airfoil to know how the flow varies with shape. The lower surface of 4412 airfoil is flat while the lower surface of redesigned airfoil is slight curve.

COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions.

LIFT (L)

Lift is an artificial force manipulated by pilot; it is generated through the wings. This is the upward component of force, acting perpendicular to the direction of Motion. The word ‘upwards’ is used in the same sense that the pilot’s head is above his feet [2].

$$\text{Lift, } L = \frac{\rho U^2 A C_L}{2}$$

Where, C_L is the lift coefficient, $\boldsymbol{\rho}$ is the fluid dynamic is the true air velocity and A is the plan form area

DRAG (D)

This is the component of force acting in the opposite direction to the line of flight. It is the force that resists the motion of the aircraft [2].

$$\text{Drag, } D = \frac{\rho U^2 A C_D}{2}$$

Where, D is the Drag force, ρ is the density of the fluid is the velocity of the object relative to the fluid, A is the reference area and C_d is the drag coefficient

METHODOLOGY

Computational fluid dynamics (CFD) is a tool used to analyze the fluid flow problem. It allows the optimization of design parameters without the need for costly experimental testing of prototype. CFD modeling process consists of first taking the real world fluid geometry and replicating this in the virtual environment. Then a mesh can be created to divide the fluid up into discrete sections. Boundary conditions must then be entered into the model to designate parameters such as the type of fluids to be modeled or the details of any solid edges or flow inlets/outlets. The simulation is then ready to be run and when a converged solution is found, it must be carefully analyzed to establish whether the mesh is appropriately modeling the flow conditions. Generally, some form of mesh refinement will be necessary to put in further detail around the areas of interest.

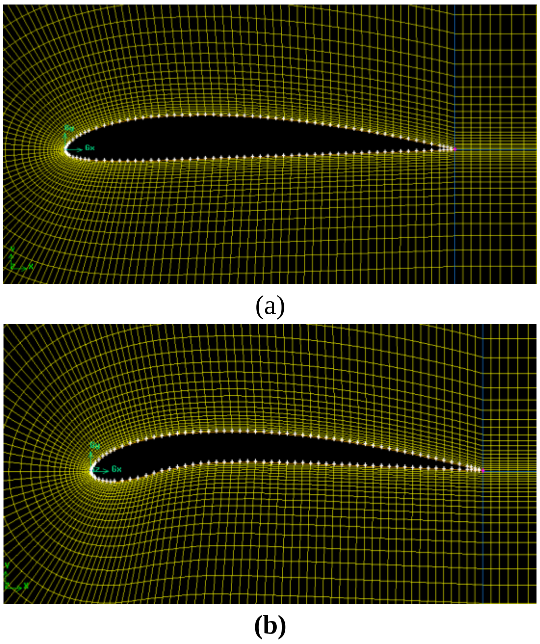

Fig.1: (a) Meshed face of 4412 airfoil and (b) Meshed face of redesigned airfoil.

RESULT AND DISCUSSION

There are many parameters such as Reynolds number, thickness ratio and angle of attack on which airfoil characteristics and performance depends. But this report is based on only angle of attack. Here, for different angle of attack lift coefficient, drag coefficient, pressure contour, and velocity vectors of 4412 airfoil are simulated and also compared with a redesigned airfoil. In this study Reynolds number is 16.7×10^5 and considered as constant.

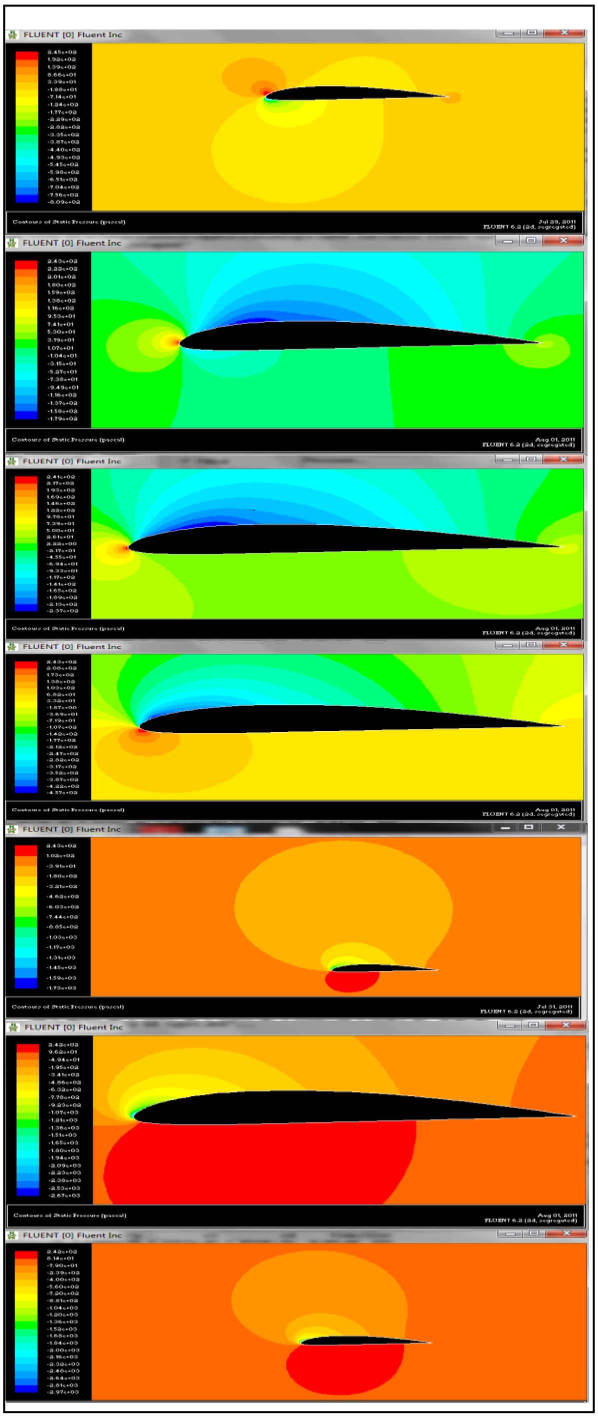

Fig.2: Contours of static pressure at -6^0, 0^0, 4^0, 8^0, 12^0, 15^0 and 16^0 respectively of NACA 4412 airfoil.

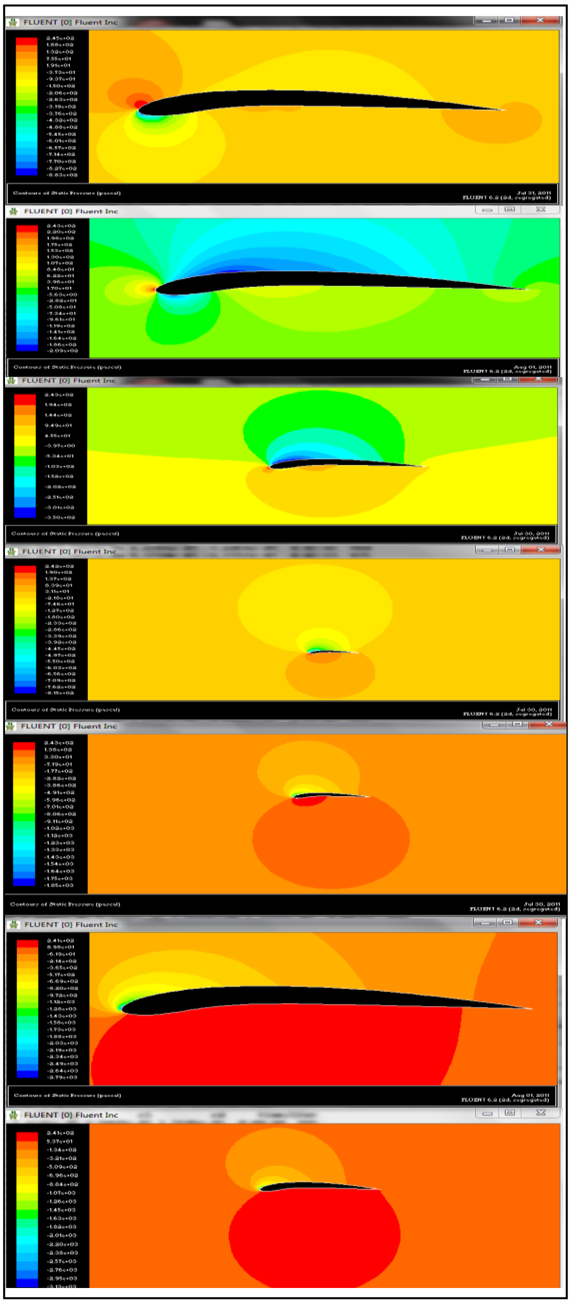

Fig.3: Contours of static pressure at -6^0, 0^0, 4^0, 8^0, 12^0, 15^0 and 16^0 respectively of Redesigned airfoil

“Fig. 2 and Fig 3 show” the static pressure contours for different angles of attack of both redesigned and NACA 4412 airfoil when the Reynolds number is 16.7×10^5. It is founded that under the same condition, the suction of upper surface of the redesigned airfoil is greater than that of the NACA 4412 airfoil and the area of the rear separation zone of redesigned airfoil is smaller than that of the NACA 4412 airfoil.

“Fig. 4 and Fig 5 show” the velocity vectors of both NACA 4412 airfoil and redesigned airfoil at different angle of attack. From the figure it is seen that, the velocity on the upper surface is faster than the velocity on the lower surface in case of both airfoil. Again, comparatively the velocity on upper surface of redesigned airfoil is faster than the velocity on upper surface of NACA 4412 airfoil. On the Leading edge, we see a stagnation point, where the velocity of the flow is nearly zero. The acceleration of fluid on the upper surface as can be seen from the change in colors of the vectors. On the trailing edge, the flow on the upper surface decelerates and converges with the flow on the lower surface and also with the increase of angle of attack the separation point moves towards the leading edge.

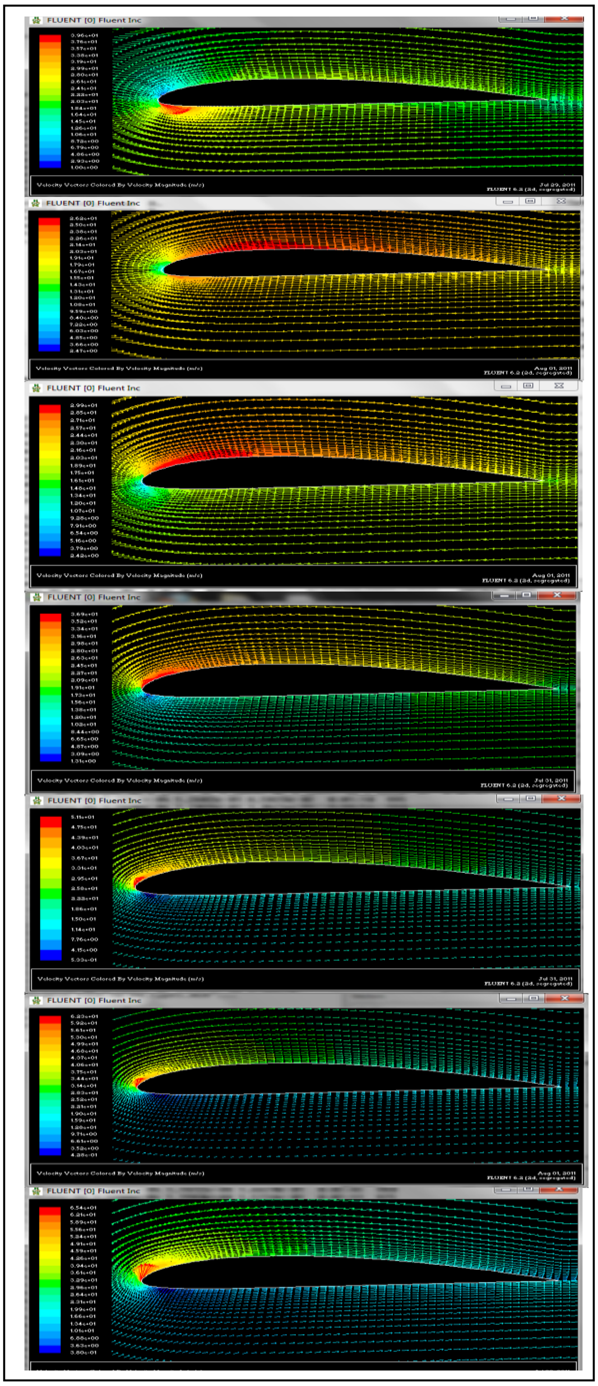

Fig. 4: Velocity vectors of NACA 4412 airfoil at -6^0, 0^0, 4^0, 8^0, 12^0, 15^0 and 16^0 respectively.

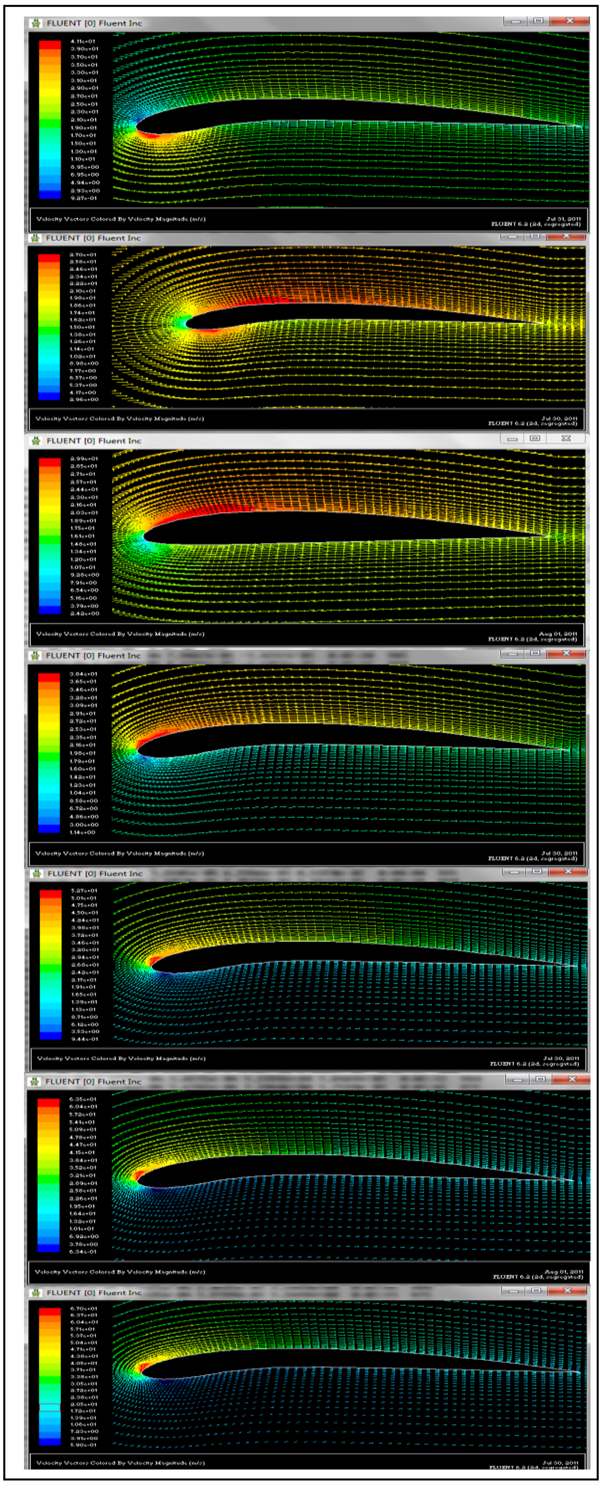

Fig. 4: Velocity vectors of redesigned airfoil at -6^0, 0^0, 4^0, 8^0, 12^0, 15^0 and 16^0 respectively.

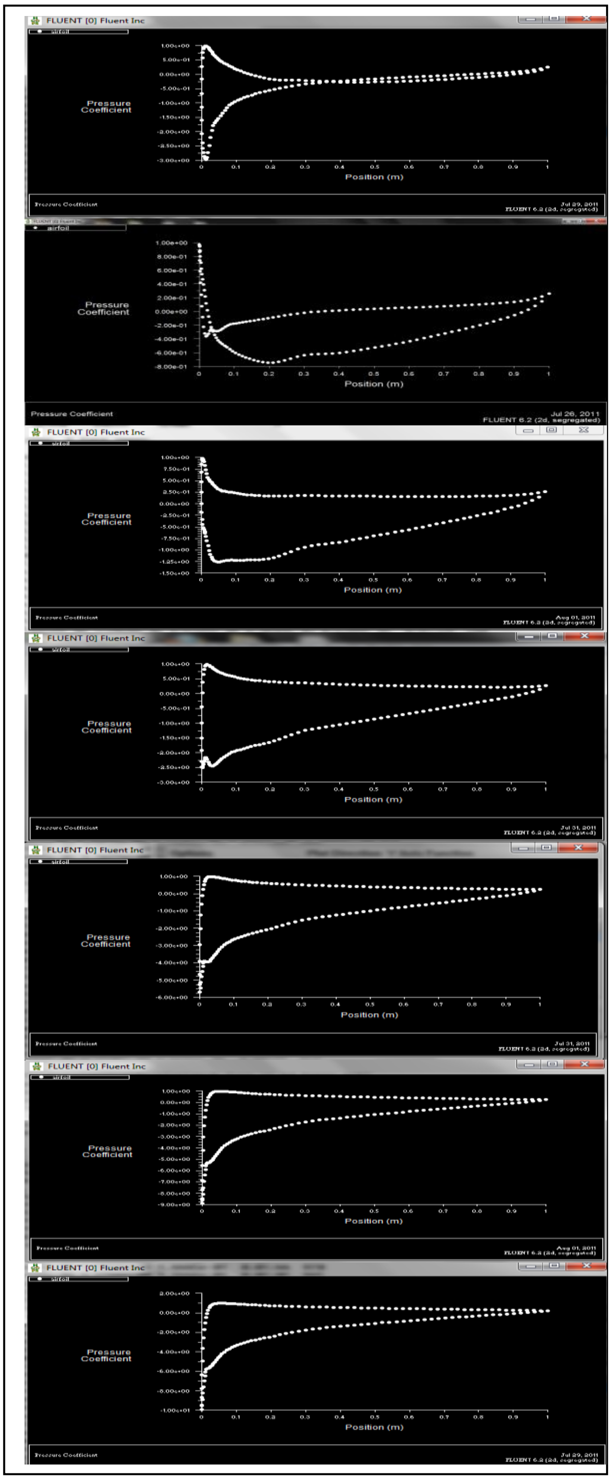

Fig. 6: Pressure coefficient vs. position curve of NACA 4412 airfoil at -6^0, 0^0, 4^0, 8^0, 12^0, 15^0 and 16^0 respectively.

"Fig. 6 and Fig 7 show" the pressure coefficient vs. position in percentage curve for NACA 4412 airfoil and redesigned airfoil respectively. The negative part of the plot is upper surface of the airfoil as the pressure is lower than the reference pressure.

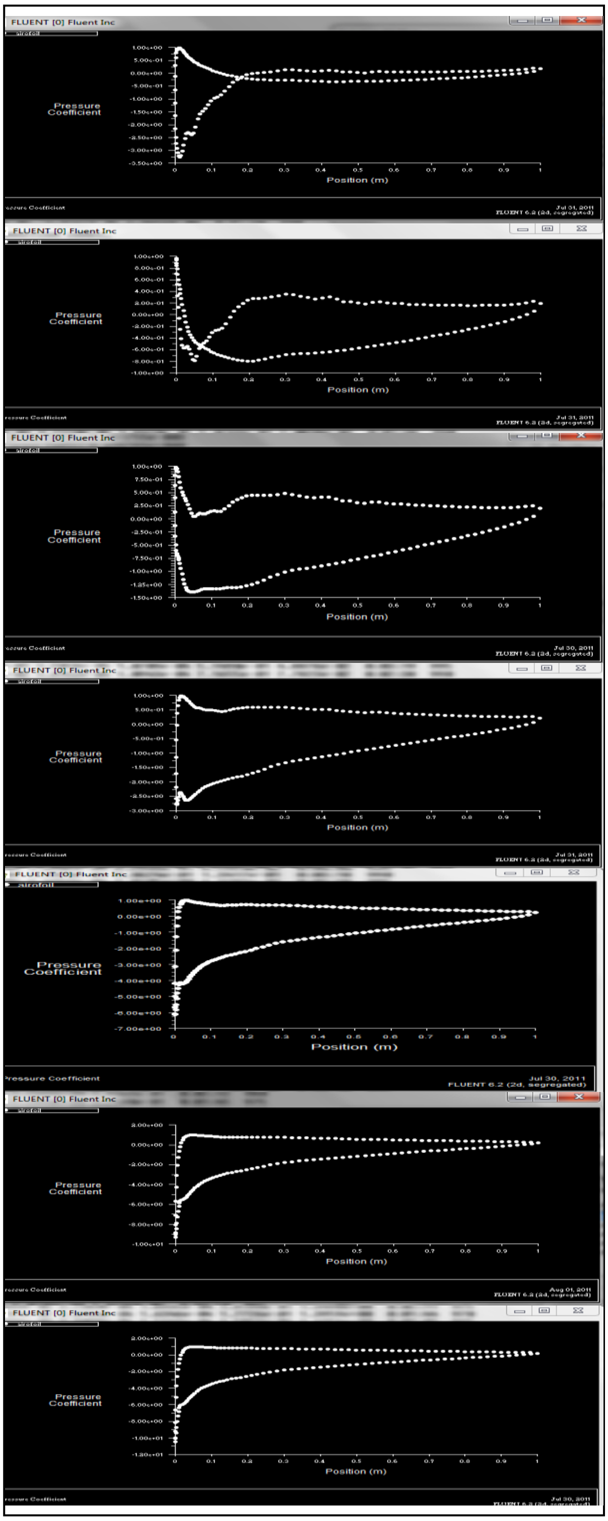

Fig. 7: Pressure coefficient vs. position curve of Redesigned airfoil at -6^0, 0^0, 4^0, 8^0, 12^0, 15^0 and

"Fig 8 show" the comparison chart of lift coefficient of NACA 4412 airfoil and redesigned airfoil. From This chart it is seen that the lift coefficient of redesigned airfoil is higher than the lift coefficient of NACA 4412 airfoil. The maximum lift coefficient of both airfoils is at 15^0.

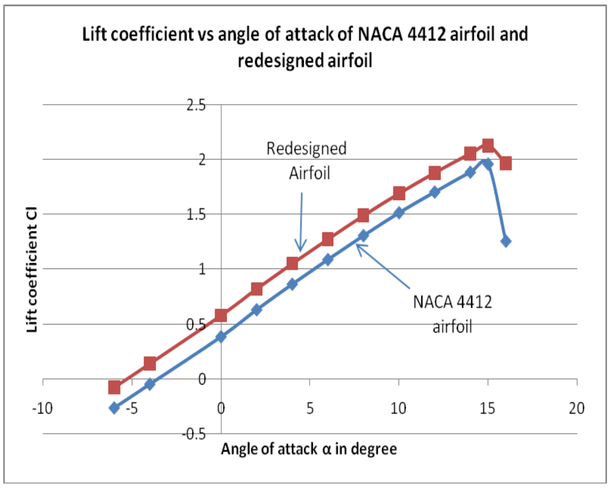

Fig.8: Comparison chart of lift coefficient of redesigned and NACA 4412 airfoil.

"Fig 9 shows" the comparison chart of drag coefficient of NACA 4412 airfoil and redesigned airfoil. From this chart it is seen that the drag coefficient of redesigned airfoil is slightly higher than the NACA 4412 airfoil. With the increase of angle of attack the separation area of the airfoil increases, the pressure on the airfoil surface increases, suction decreases, the elevating force decreases and drag force increases.

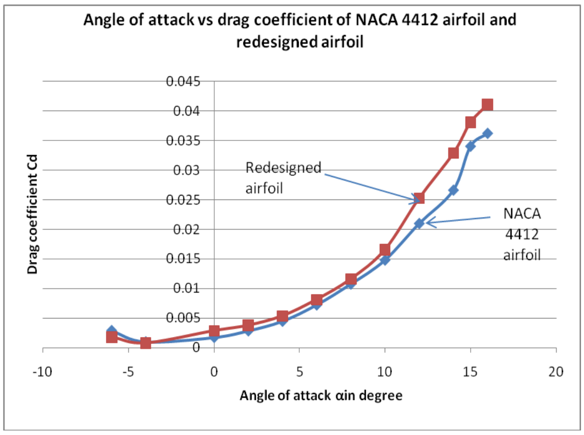

Fig.9: Comparison chart of Drag coefficient of redesigned and NACA 4412 airfoil.

"Fig 10 show" the comparison chart of lift coefficient to drag coefficient ratio of both NACA 4412 airfoil and redesigned airfoil. The maximum lift to drag coefficient ratio of NACA 4412 airfoil is higher than the maximum lift to drag coefficient ratio of redesigned airfoil and also in case of NACA 4412 airfoil the point of maximum lift to drag coefficient ratio is before 2^0 where in case of redesigned airfoil the point is at 2^0.

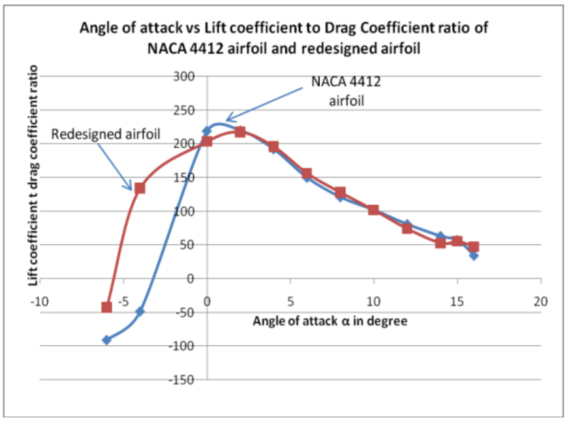

Fig 10: Comparison chart of lift to drag coefficient ratio of redesigned and NACA 4412 airfoil.

DATA TABLE

	NACA 4412 airfoil		Redesigned airfoil	
Angle of attack atattatta	Lift coefficient Cl	Drag coefficient Cd	Lift coefficient Cl	Drag coefficient Cd
-6^0	-0.2644	0.002915	-0.07626	0.00181
-4^0	-0.0484	0.00100	0.14125	0.000774
0^0	0.38427	.001743	0.58082	0.002851
2^0	0.63032	0.00286	0.82233	0.003773
4^0	0.86361	0.00448	1.0525	0.00536
6^0	1.088	0.00724	1.274	0.008132
8^0	1.3064	0.01076	1.4899	0.011608
10^0	1.5154	0.01481	1.6918	0.016567
12^0	1.7037	0.021	1.8795	0.025258
14^0	1.8848	0.02663	2.0541	0.032862
15^0	1.9588	0.03404	2.1281	0.038074
16^0	1.256	0.03623	1.9652	0.041099

CONCLUSION

From the above study it can be that the computational fluid dynamics is very important for our modern world. Computational study of fluid dynamics not only reduces the cost and time but also helps us to analyze the fluid flow problem in hazards condition very easily. This project of computational study of fluid around NACA 4412 airfoil and comparison of result with redesigned airfoil concluded the followings-

- The lift coefficient increases with the increase of angle of attack and after certain angle lift coefficient start to decrease.
- Lift coefficient is maximum at15^0 angle of attack for both airfoils. I.e stalling angle is 15^0.therefore, the shape of the airfoil does not vary stalling angle largely.
- Lift coefficient is greater for redesigned airfoil but lift to drag ratio is higher for NACA 4412 airfoil.

REFERENCES

[1] Dmitri Kuzmin, A Guide to Numerical Methods for Transport Equations, 2010

[2] Fifth Edition, E.L. Houghton and P.W. Carpenter ,Professor of Mechanical Engineering, The University of Warwick, Aerodynamics for Engineering Students.

NOMENCLATURE

Symbol	Meaning	Unit
L	Lift	(N)
D	Drag	(N)
ρ	Density	Kg/sec
C_d		
C_l	Drag coefficient	
A	Lift coefficient	(m2)
U	Area	m/sec
	Free stream velocity	