International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 9, November- 2012

Flow And Heat Transfer Characteristics Over Naca0018 Aerofoil

M. Sri Rama Murthy*
Associate professor
Department of mechanical engineering,
Sir C.R.R.College of Engineering,
Eluru-534007, A.P, India.

*Corresponding author

A.V.S.S.K.S. Gupta
Professor,
Department of mechanical engineering,
JNTUH College of engineering,
Hyderabad – 72, A.P., India.

ABSTRACT

Experiments are carried out to study heat transfer and drag variations over NACA0018 aerofoil. The airfoil is tested for cold and hot temperature conditions in a subsonic wind tunnel. In this work, the flow and heat transfer characteristics are studied at different flow speeds and angles of incidence. Drag force has been measured for both the conditions i.e. cold and hot. The results have been presented in the form of drag force, drag coefficient at different Reynolds numbers and angles of incidence. It is observed that that drag force and drag coefficients are larger at higher angles of attack and the drag force increases at high Reynolds number while the drag coefficient decreases. The drag coefficient values are reduced for heated airfoil when compared with the cold airfoil. The heat transfer experiments have been carried out under constant heat flux condition. Heat transfer coefficients are determined from the measured wall temperature and ambient temperature and presented in the form of Nusselt number. The variation of Nusselt number has been shown with Reynolds number for different angles of attack.Nusselt number increases as the flow speed increases and heat transfer rate is higher at the angle of attack is 30º and it is less when the angle of attack is 0º. It is found that the larger the angle of incidence the larger is the drag coefficient and higher the heat transfer rate.

KEY WORDS: Angle of attack, Drag coefficient, Reynolds number, Nusselt number, Boundary layer separation.

INTRODUCTION

Over the last few decades extensive research work is going on the study of flow characteristics over different types of aero foils. The flow behavior becomes increasingly complex depending upon the shape and the free stream flow conditions. In the past, different configurations were tested for different operating conditions. The computational and experimental investigations were carried for low Reynolds number flows over SD7003 aerofoil to predict transition by Yvan et. al. [1]. The effects of increase of Reynolds number [Re] on clean and rough aero foils were investigated by Freudenreich et. al. [2]. From this study it was found that the clean surface configuration shows no drastic loss in max lift, the lift-to-drag ratio decreased from 95 to approximately 85, mainly due to an increase of drag. In contrast, to that rough surface configurations showed increased performance for increase of Reynolds number. Fleming and Taylor [3] developed a computer model for simulation of incompressible turbulent flow and heat transfer over rough surfaces using discrete element method. In one case roughness included flow separation was indicated. Effect of regular surface perturbations on flow over an aerofoil was studied by Aravind Santhana Krishnan and Jacob [4]. In this, smoke wire flow visualization was performed for qualitative observation of the separation region in both the perturbed and unmodified aerofoil geometry cases.

Experimental investigations were carried out by Chenna Raju and Viswanath [5] to study the effect of riblets on the base drag of GAW aerofoil at low speeds. Results show that riblets provide base drag reduction of about 0.7% of the total drag, in addition to significant viscous drag reduction. A large Eddy simulation of the flow around a NACA0012 aerofoil at a Reynolds number of 5, 00,000 is presented by Marsden et. al. [6]. In this study LES showed a well placed transition zone and turbulence levels in good agreement with experimental data. Experiments were carried out to study the effects of various ice accretions on the aerodynamic performance of a 36 inch chord 2D business jet aerofoil by Addy et.al. [7]. From this, it was observed that ice shapes resulted in as much as 48% reduction in max lift coefficient from that of the clean aerofoil. Large increase in drag and changes in pitching moment were also observed.

Jamushevskis et.al. [8] developed a method for optimization of subsonic aerodynamic shape by using the metamodeling approach. The method is demonstrated by the solution of a test problem for an aerofoil subsonic shape optimization. Julian Allen & Look [9] developed a model for determining the chord wise distribution of the rate of heat transfer from the surface of a wing or body of revolution to air.

A CFD based study has been conducted by Patil et.al. [10] to carry out Reynolds averaged Navier Stokes (RANS) based computations to investigate the aerodynamics of Micro Air Vehicle (MAV) at various angles of attack. It enlightens the requirements for the proper meshing of the computational domain, discuses computational aspects and presents results of the numerical study to provide insights into the MAV aerodynamics.

Gorji – Band by et.al. [11] performed the Numerical solution of flow and heat transfer on staggered arrangement circular tubes with aerofoil vortex generator and modeled by Navier –Stokes and energy equations. The results depicted that the usage of aerofoil obstacle is efficient for increasing heat transfer in spite of more friction coefficient and more pressure reduction.

The effect of aspect ratio on the aerofoil performance was investigated for airflow about axially symmetric wings as a function of the angle of attack by Kopac et. al. [12]. Three different types of airfoils were tested under the airflow speed of 33.76m/s and it is concluded that the aerofoil, with the aspect ratio of 2.761 yields the optimum performance.

Experimental investigations in fluid flow and heat transfer have been carried out to study the effect of wall proximity due to flow separation around rectangular prisms by Dipes Chakrabarty [13]. The pressure distribution shows positive values on the front face whereas on the rear face negative values of the pressure coefficient have been observed. The drag coefficient decreases with the increase in angle of attack as the height ratio decreases. Heat transfer experiments have been carried out under constant heat flux condition. The variation of average Nusselt number has also been shown with different angles of attack.

Experimental investigations and simulation studies in fluid flow and heat transfer have been carried out by Nabhan [14] for the aerofoil made from aluminum alloy6014-T4.From this study he has concluded that temperature distribution and heat transfer rate was changed when angle of attack changed.

From the various literatures, it appears that the experimental investigations on fluid flow and heat transfer studies over an aerofoil for different temperature conditions (i.e. cold and hot) with different angles of attack and Reynolds numbers has not been adequately covered. In the present investigation an attempt has been made to study the flow characteristics over aerofoil for different temperature conditions i.e. hot and cold condition. In addition, it has been tried to determine the coefficient of drag at different flow speeds and foil tilt positions for both the conditions. Further attempt has been made to determine the heat transfer coefficients for different Reynolds numbers and angles of attack under constant heat flux boundary condition. Also investigations are carried out to study the relationship between Nusselt number and Reynolds number for different angles of attack.

MATERIALS AND METHODS

Experiments are carried out in a wind tunnel and the details are as follows :

Altech open circuit wind tunnel is designed for use in student Engineering laboratories and other industrial and Government research facilities. This Wind

Tunnel can be used to study the pressure distribution and lift drag characteristics of airfoils, cylinder etc. Reynolds number up to 25, 00,000 can be achieved with this tunnel. The wind tunnel used for experimentation is shown in the figure1.

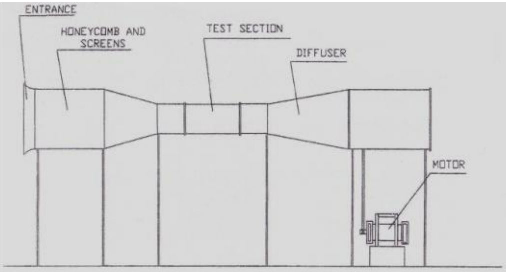

Figure 1: Experimental Setup

The wind tunnel is of suction type with an axial flow fan driven by a variable speed DC motor. It consists of an entrance section with a bell mouth inlet containing a flow straightener, screens and a straw honey comb. This section is followed by a 6.25:1 contraction section, the test section, a diffuser and the duct containing the axial flow fan. The whole unit is supported on steel frames. The complete wind tunnel except the test section is constructed of mild steel iron sheets for strength and rigidity. The test section is made of teak wood and has glass window for visual observation of flow phenomena. The control of the DC motor is by a rectifier controlled variable speed drive. The experiment has been carried out in subsonic wind tunnel with a test section 300 mm high 300mm wide and 800 mm long. The aerofoil has been fixed along the width of the test section.

AIR FOIL DETAILS

A test model of NACA 0018 aerofoil has been selected for this study. The aerofoil is made of aluminum material with a chord of 16 cm and a span of 25 cm .The figure2show the details of aerofoil and thermocouples at twelve different locations on the surface of aerofoil, which are used to measure the local wall temperature distribution at steady state condition for calculation of heat transfer coefficient and Nusselt number.

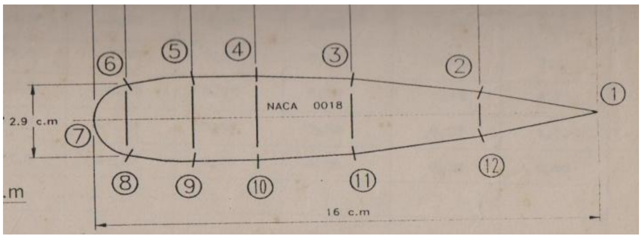

Figure 2: Aerofoil

To study the flow characteristics on the airfoil, the airfoil is tilted at different angles of attack namely 0, 10, 20, 30 degrees. This is done by adjusting the wind tunnel tilt mechanism where the airfoil is attached. In this study, the flow characteristics are investigated experimentally for a range of Reynolds numbers from $6x10^4$ to $20x10^4$. Measurement of free stream air velocity is performed using a Pitot tube and with a linkage mechanism and a digital component force measuring transducer to determine drag force. The foil is heated using a heating element which is wrapped on the surface of the foil. Surface temperatures at various locations on the foil are measured using twelve thermo couples.

RESULTS AND DISCUSSION

Fluid Flow Characteristics

In the present study, a cold and heated airfoil is used in order to test the effect of flow characteristics on drag over the airfoil at different angles of attack and Reynolds numbers.

The drag coefficient values are determined using the following relations. The airflow velocity is determined from Pitot Tube using the equation

$$V=13\sqrt{q}, \qquad (1)$$

Where q = difference of manometer reading in cm.

And Reynolds number is calculated using the relation

$$R_e = \rho V L_C/\mu \qquad (2)$$

The coefficient of drag, C_D is determined using the following relation for the airfoil at different velocities, V and angles of attack α

$$C_D=2F_D/\rho AV^2 \qquad (3)$$

The drag force and drag coefficients at different angles of attack with different Reynolds numbers are shown in the figures 3 & 4.

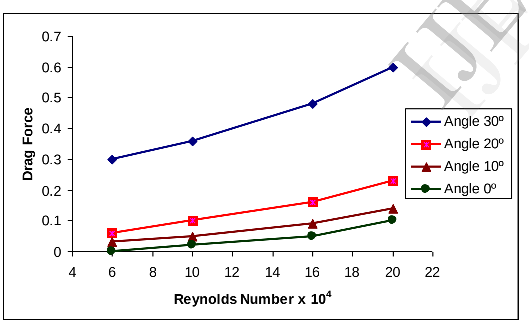

Figure 3: Drag force with Reynolds numbers at different angles of attack for cold airfoil.

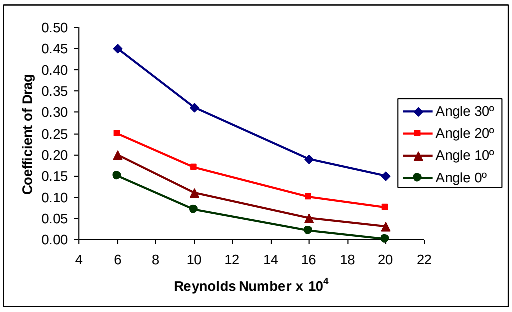

Figure Figure 4: Coefficient of drag with Reynolds numbers at different angles of attack for cold aerofoil.

The relation between drag coefficient and Reynolds numbers for the heated airfoil is shown in the figure 5 for different angles of attack.

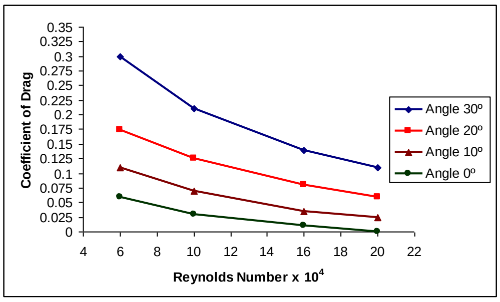

Figure 5: Coefficient of drag with Reynolds numbers at different angles of attack for heated airfoil.

Figures 3 and 4 show the drag force and drag coefficient variations on the unheated air foil with Reynolds Number at different angles of attack. It is observed that drag force and drag coefficients are larger at higher angles of attack and the drag force increases at high Reynolds Number while the drag coefficient decreases. This behavior can be explained as follows. Since the boundary layer separation is not a factor at low Reynolds Number, the pressure drag is less. Whereas viscous drag increases considerably at the surface, simply because of the increased surface area over which the frictional stresses act. Thus total drag is actually increased by stream lining at low Reynolds number. Changes in the angles of attack alter the pressure distribution, particularly on the upper surface. As the angles of incidence increases, the main flow separates from the upper surface because the shape downstream of the foil shoulder is such as to produce a severe rate of pressure rise leads to boundary layer separation and consequently to a larger pressure drag. Then total drag is increased greatly and lift suddenly decreases.

In the first part of the investigation, the relationship between the coefficient of drag and Reynolds number at different speeds is obtained without adding a heat source, while in the second part, the heat source is added to the experiment. Figure 5 shows the relationship between drag coefficient and Reynolds number at different angles of attack for heated air foil. The boundary condition adopted here is the constant heat flux. Comparing these results with the results obtained for cold airfoil, the difference in values of drag coefficient is noted. The drag coefficient values are reduced for heated air foil when compared with the cold airfoil. This is related to the thermal boundary layer thickness which increases by the increase in the supplied heat rate. A hot source also reduces friction. It is also found that the lowest drag coefficient is obtained at zero angle of incidence.

Heat transfer Characteristics

Newton's law of cooling is applied for convective heat transfer across the airfoil to determine the Nusselt number, N_u at different Reynolds numbers and angles of incidence by applying a constant heat source to the foil. Temperatures on the surface of the foil are noted down using thermo couples at 12 points longitudinally along the chord.

Mean temperature can be calculated using the formula

$$T_m = \sum_{n=i}^{n=12} T_i / n \qquad (4)$$

Neglecting radiating heat loss from the surface of the airfoil, the convective heat transfer coefficient is calculated using the formula $h = \frac{Q}{A\Delta T}$

(5)

Where Q is the heat supplied, A is the projected area of the airfoil surface and ΔT is the temperature difference between the foil surface and the surroundings. Then Nusselt number, N_u is calculated from the equation

$$N_u = \frac{L_c h}{k} \quad (6)$$

The results obtained after the application of heat source is shown in the figure 6.

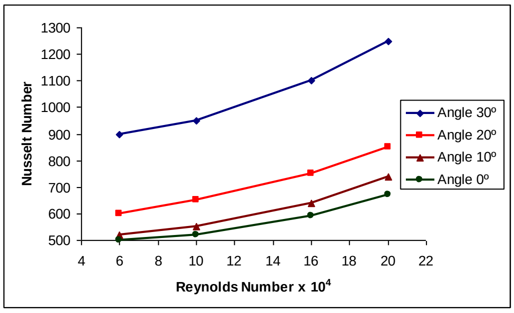

Figure 6: Nusselt number variations with Reynolds number at different angles of attack.

Figure 6 shows the Nusselt Number variations with the Reynolds numbers at constant heat source. It is observed that heat transfer rate increases as the flow speed increases. Further, heat transfer rate is higher at the angle of attack is 30^0 and it is less when the angle of incidence is 0^0. In front of the body and behind it, there is a recirculation flow, which is separate from the main flow stream. It can be seen that when the angle position increases, the recirculation flow domain increases, which in turn increases the heat transfer rate from the surface.

CONCLUSIONS

The flow and heat transfer characteristics of NACA 0018 aerofoil were investigated experimentally .The angle of attack is varied in the range of $0^\circ \le \alpha \le 30^\circ$ over the $6\text{x}10^4 \le$ Reynolds number $\le 20\text{x}10^4$.

The dependency of the drag coefficient on the angle of attack and Reynolds number is quite clear from the results .From the flow characteristics, it is concluded that drag coefficient decreases as the flow speed increases and the coefficient of drag increases as the angle of incidence increases for the same speed. And drag coefficient values are lower for hot aerofoil when compared with cold aerofoil.

From the heat transfer characteristics, it is concluded that Nusselt number increases as the Reynolds number increases. As the angle of incidence increases, the increase in Nusselt number takes place. Highest heat transfer enhancement takes place when $\alpha = 30^o$.

NOMENCLATURE

A Projected area of Airfoil, m^2

C_D Drag coefficient

F_D Drag force, N

h Heat transfer coefficient, $w/m^2 K$

k Thermal conductivity of air, w/m k

L_c Characteristic length, m

N_u Nusselt number

n No of Thermo couples

q Difference of manometer reading, cm

Q Convective heat transfer rate, watts

R_e Reynolds Number based on velocity of air and characteristic length of bluff body

T_m Mean airfoil surface temperature, K

T_i Thermo couple temperature, K

ΔT Temperature difference between foil surface temperature and the surroundings, K

V Fluid flow Velocity, m/sec

α Angle of attack, degrees

ρ Density of air, kg/m^3

μ Viscosity, Ns/m^2

REFERENCES :

[1] Yvan, et.al., Computational and experimental investigations low-Reynolds's number flows past an aero-foil, The Aeronautical Journal, January 2007, paper No.3109, pp.17-29.

[2] Freudenreich,K., Kaiser., Scaffarcyk,A.P., Winkler,H., and Stahl,B., Reynolds number and roughness effects on thick airfoils for wind Turbines, Wind Engineering, vol.no.28, 5, 2004, pp. 529-546.

[3] Fleming,K.R., and Taylor,R.P., Incompressible Naviour Stokes Algorithm for flow & Heat transfer over rough surfaces, AIAA 27th thermo physics conference, July-6-8, 1992, AIAA – 2925.

[4] Arvind Santhana Krishnan and Jacob, J.D., Effect of regular surface perturbations on flow over an Airfoil, AIAA, pp.1-9.

[5] Chenna Raju & Viswanadh, P.R., Base drag reduction due to riblets on a GAW (2) Airfoil, NAL, Bangalore.

[6] Olivier Marsden., Christophe Bogey., Christophe Bailly., Direct Noise Computation around a 3-D NACA 0012 airfoil, 12th AIAA aero acoustics conference, 8-10 may 2006, Cambridge.

[7] Harold E.Addy., et.al., A Wind Tunnel Study of Icing effects on a business Jet Airfoil, NASA, 2003-212124.

[8] Jamshevskis, A., Optimization of subsonic Aerodynamic shape by using metamodeling approach, 5th International conference on "Industrial Engineering" 20-22nd April 2006, Estonia.

[9] Julian Allen. and bonne C.Look., A method for calculating Heat transfer in the Laminar flow region of bodies, NACA, Report no.764, pp. 178-182.

[10] Patil,D.J., Khan,M., and Gupta,B.K., Numerical simulations of Low Reynolds Number Flow around Micro Air vehicle, proceedings of INCAST, 26-28 June 2008, 054.

[11] Gorji – Bandpy,M., Soleimam,S., and Hossein – Nejad,F., Pressure and Heat transfer in staggered arrangement circular tubes with Airfoil Vortex Generator, International conference on Sustainable development, 12-14 december 2007.

[12] Kopac,M., yilmaz,M., Gultop,T., An investigation of the Effect of Aspect ratio on the Airfoil performance, American Journal of Applied Science, 2(2), 2005, pp. 545-549.

[13] Dipes Chakrabarty, Experimental Investigations on fluid flow and heat transfer from a rectangular prism approaching the wall of a wind tunnel, Dept. of Mechanical Engineering, Konkuk University, South Korea.

[14]Md.B.W.Nabhan.,Heat transfer and fluid flow characteristics over streamlined symmetrical body positioned at unfavorable angles to the flow,ISESCO science and technology vision,vol.6,November2010(2-8).