Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 105 (2015) 329 – 334

Procedia Engineering

www.elsevier.com/locate/procedia

6th BSME International Conference on Thermal Engineering (ICTE 2014)

Numerical investigation on the delay of boundary layer separation by suction for NACA 4412

R. Azim[a]*, M. M. Hasan[a] and Mohammad Ali[b]

[a]*Department of Mechanical Engineering, Bangladesh University of Engineering & Technology (BUET),Dhaka-1000.*

[b] *Professor,Department of Mechanical Engineering, Bangladesh University of Engineering & Technology (BUET),Dhaka-1000.*

Abstract

Transition flow over airfoils at the higher angle of attack shows a lot of unsteady phenomena such as local separation regions, boundary layer transition, turbulence and shock boundary layer interaction. These phenomena are associated with high energy loss and adversely effects the aerodynamic loads in the form of lift loss and drag increase. Controlling the flow through separation delay by suction at different slots, by flaps, by introducing bumps and sophisticated high lifting devices can mitigate the aerodynamic losses. This paper focuses on the delay of boundary layer separation of 2D NACA 4412 by suction using CFD analysis. Picking out the right suction position augments the aerodynamic performance. So a slot with a width of 2% of the chord length is placed at five different locations starting from 48% to 70% of the chord length. The main part of the paper is related to the selection of a suction position and outcome of different suction pressures at a definite slot. Suction with the lower pressure at a definite position moves the separation of boundary layer in the vicinity of trailing edge of the airfoil most. By suing suction at suction pressure 65kPa on 68% of the chord length of the airfoil with a constant angle 2^0 with the upper surface of the airfoil, AOA=13^0 and M=0.6, it is possible to move the transition to turbulent flow about 91% of the chord length of the airfoil near the trailing edge where it is found at 43% of the chord length of the airfoil without suction. So the laminar region is extended and the lift increases. Along with this, at low angle of attack, the lift to drag ratio after suction increases about 2.24 times compared to that of without suction.

Peer-review under responsibility of organizing committee of the 6th BSME International Conference on Thermal Engineering (ICTE 2014)

Keywords: *Flow control ; Laminar flow ; Turbulent flow ; Lift and drag force ; Boundary layer separation ; Pressure co-efficient*

Nomenclature and abbreviation

M	Mach number
AOA	Angle of attack

* Corresponding author. Tel.: +8801911560549
E-mail address: riasatme08@gmail.com

1877-7058
Peer-review under responsibility of organizing committee of the 6th BSME International Conference on Thermal Engineering (ICTE 2014)
doi:10.1016/j.proeng.2015.05.013

NACA	National Advisory Committee for Aeronautics

1. Introduction

The turbulent flow separation around the trailing edge (TE) of an airfoil affects the aerodynamic performances (like lift reduction, drag enhancement) very severely. At the higher angle of attack, the effect of adverse pressure gradient enhances the formation of separation wakes around trailing edge of the airfoil. The application of suction at proper position greatly subjugates the stream wise momentum loss in the wake. For the past years, considerable effort had been devoted to the investigation of the application of suction applied either through open slit or porous wall strip for the purpose of reduction of the skin friction drag or for the control of the boundary layers. Braslow [1] illustrated the history of suction type laminar flow control. Seifert et al. [2] investigated unsteady suction and blowing on a symmetric airfoil to increase post-stall lift. Two different means of suction had been investigated in the past: discrete suction through slots and distributed suction. Boermans [3] had shown that discrete suction allowed an abrupt pressure increase at the location of the slot. Eppler [4] found that the design case of an airfoil with distributed suction specifies together with the suction compartments and the suction pressure, the porosity of the surface. Oyewola et al. [5] studied that the effects of localized double suction applied through a pair of porous wall strips on a turbulent boundary layer had been quantified through the measurements of mean velocity and Reynolds stresses. Richards et al. [6] had shown that, it was impossible to maintain laminar flow aft of the suction slot at high Reynolds numbers because of the dynamic instability of the laminar layer over the concave surface. So the porosity of the surface of airfoil and geometry of the suction compartments are determined by the designed case. That’s why, it is considered that many off-design cases like surface porosity and suction compartment geometry are fixed and the pressure in the suction slot is the only variable for numerical investigation over an airfoil like NACA 4412.

2. Numerical procedures

Navier–Stokes equations arise from applying Newton's second law to fluid motion, together with the assumption that the fluid stress is the sum of a diffusing viscous term (proportional to the gradient of velocity) and a pressure term - hence describing the viscous flow. In an inertial frame of reference, the general form of the equation is:

$$\rho\left(\frac{\partial v}{\partial t} + v.\nabla v\right) = -\nabla p + \nabla.T + f$$

Where, $\boldsymbol{v}$ is the flow velocity, ρ is the fluid density, p is the pressure, T is the stress tensor, f represents body forces (per unit volume) acting on the fluid and ∇ is the Del operator.

Numerical flow simulation is performed by solving Navier-Stokes equations, which are formulation of mass, momentum and energy conservation laws.

The Spalart Allmaras [7] turbulence model solved a modelled transport equation for kinematic eddy viscosity without calculating the length scale related to the shear layer thickness. The variable transported in the Spalart Allmaras model is $\tilde{v}$ which is assimilated, in the regions which are not affected by strong viscous effects such as the near wall region, to the turbulent kinematic viscosity. This equation has four versions, the simplest one is only applicable to free shear flows and the most complicated, which is written below, can treat turbulent flow past a body with laminar regions.

$$\frac{\partial}{\partial t}(\rho\tilde{v}) + \frac{\partial}{\partial x_i}(\rho\tilde{v}\, u_i) = G_v + \frac{1}{\sigma_{\tilde{v}}}\left[\frac{\partial}{\partial x}\left\{(\mu + \rho\tilde{v})\frac{\partial\tilde{v}}{\partial x}\right\} + C\,\rho\left(\frac{\partial\tilde{v}}{\partial x_i}\right)^2\right] - Y_v + S_{\tilde{v}}$$

This transport equation brings together the turbulent viscosity production term, G_v and the destruction term, Y_v. The physics behind the destruction of turbulence occurs in the near wall region, where viscous damping and wall blocking effects are dominants. No heat generation or transfer is considered. The other terms or factors are constants calibrated for each physical effect which needs to be modelled. This equation allows to determinate $\tilde{\boldsymbol{v}}$ for the computation of the turbulent viscosity, μ_t from:

$$\mu_{t=}\,\rho\tilde{v}\,f_{v1}$$