Advances in Aerospace Science and Applications.
ISSN 2277-3223 Volume 3, Number 2 (2013), pp. 75-82

http://www.ripublication.com/aasa.htm

Numerical Investigation of Vicous Drag Reduction Using Riblets

**Dinesh Kumar G[1], Vignesh Kumar R[2], Ajay Kumar.M[3],
Srikanth J.V.R[4] and Soorya Prasad[5]**

[1]*Assistant professor, School of Aeronautical Sciences, Hindustan University.*
[2345]*Aeronautical Engineering, Hindustan University, Chennai, INDIA.*

Abstract

This paper explains the performance improvement of drag reduction using computatational investigation with the help of riblets in an aerofoil. Riblets, are micro-grooves carved on the surface of the wing and aligned to the freestream direction .In order to improve the drag reduction, riblets was placed above the wing by which drag is reduced on streamlined aero foil. The task seems to be daunting as the wing performance decreases with the higher altitude due to drop in flow of air, this draw back can be encountered by placing the riblets at the exact position on the wing surface as it has the capability to produce better flow of air even at higher altitude.

The selection of Riblets is based on numerous calculations at various conditions. Numerical analysis of the wing with and without the riblets were performed The inference of the CFD investigation of flow over riblets reduces the viscous drag of the wing at low angle of attacks i.e. even at the angle of attack below 5.

Keywords: Drag reduction, Riblets, Numerical Analysis, CFD, Angle Of Attack.

1. Introduction

An aerofoil is a shape of a wing or blade that causes the aerodynamic forces. Forces acting in a plane are Lift, Drag, Thrust and Weight, of which aerofoil is responsible for Lift and Drag. Lift acts upwards perpendicularly to the direction of forward motion and Drag acts parallel along the direction of flow. The characteristic of an aerofoil is that to attain maximum lift and reduce drag. Our aim in this paper is to reduce drag by using riblets. Riblets, which are micro-grooves on the surface and aligned to the freestream direction, have been studied most extensively and the results from various studies have been sufficiently promising and encouraging that the concept has been evaluated in flight tests.

1.1 Types of Drag

1. Induced Drag:
2. Parasite Drag:
3. Skin Friction Drag:
4. Form Drag:
5. Interference Drag:

Total Drag= Induced drag + Parasite drag.

1.2 Drag Reduction Using Riblet Film Applied to Airfoils

In a literature survey, Robert W. Deters, y Steven P. Henry,z and Michael S. Seligx University of Illinois at Urbana-Champaign, Urbana, IL 61801, USA explained that results of a study that was commissioned by the 3M Renewable Energy Division to measure the drag reduction by using riblet film on airfoils specially designed for wind turbine applications. At angles of attack spanning the low drag range of the airfoil. Tests were run with riblet covering different sections of the airfoil in order to determine the optimal riblet location in terms of drag reduction. Results showed that the magnitude of drag reduction depended on the angle of attack, Reynolds number, riblet size, and riblet location. For some configurations, riblets produced significant drag reduction of up to 5%, while for others riblets were detrimental. Trends in the results indicated an optimum riblet size of 62-m for the range of Reynolds numbers at which tests wereconducted. The airfoil chord was 18 in (0.457 m). Results also showed that eachriblet size performed best at a given Reynolds number with the optimal Reynolds number decreasing with an increase in riblet size.

2. Computational Fluid Dynamics

Any fluid flow in the universe is governed by a set of equations.

1. Continuity Equation.

$$\frac{\partial}{\partial t}\iiint_V \rho dv + \iint_S \rho V.dS = 0$$

2. Momentum Equation.

$$\frac{\partial}{\partial t}\oiiint_V \rho V\, dV + \oiint_S (\rho V.dS)V = -\oiint_S p\, dS + \oiiint_V \rho f dV + F_{viscous}$$

3. Energy Equation.

$$\oiiint_V q\rho dV + Q_{viscous} - \oiint_S pV.dS + \oiiint_V \rho(f.V)dV + W_{viscous} = \frac{\partial}{\partial t}\oiiint_V \rho\left(e + \frac{V2}{2}\right)dV + \oiint_S \rho\left(e + \frac{V2}{2}\right)V.dS$$

3. Meshing

In order to analyze fluid flows, flow domains are split into smaller subdomains (made up of geometric primitives like hexahedra and tetrahedral in 3D and quadrilaterals and triangles in 2D). The governing equations are then discretized and solved inside each of these subdomains. Typically, one of three methods is used to solve the approximate version of the system of equations: finite volumes, finite elements, or finite differences.

3.1 Methodology of the Project

We have done a literary survey on the drag reduction techniques and finalized by selecting the riblets used for drag reduction. The next step we did selection for the wing used for the experiment and modeled the wing using geometry in ANSYS Wing Modeler. CFD Analysis is carried out for various Angle of Attacks for velocities 50m/s.

4. Modeling & Analysis

The modeling of the models is done using the ANSYS Design modeler software. The Description of the Steps is give below.

Aerofoil used: Root Aerofoil of B29 Aircraft
Chord Length: 5.18 m
Wing Span: 10 m

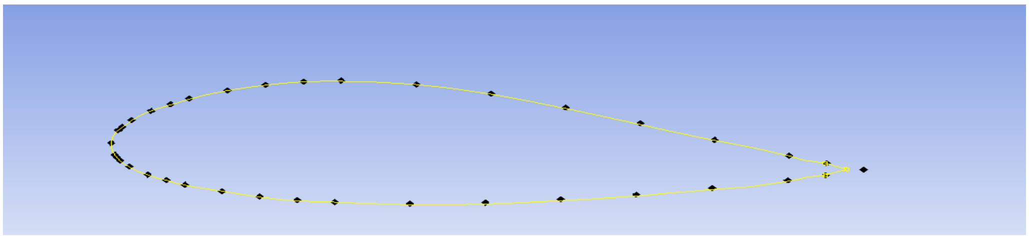


Figure1: The points imported in to the Design Modeler.

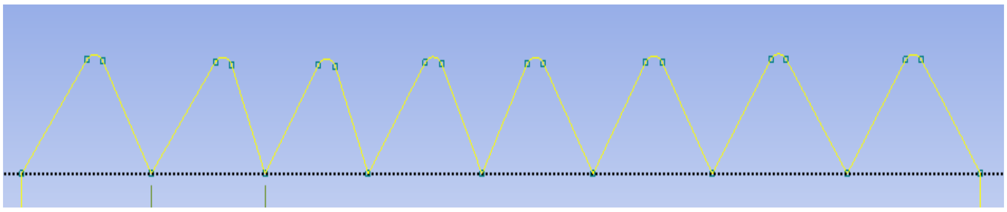


Figure2: Shape of the Riblet Profile.

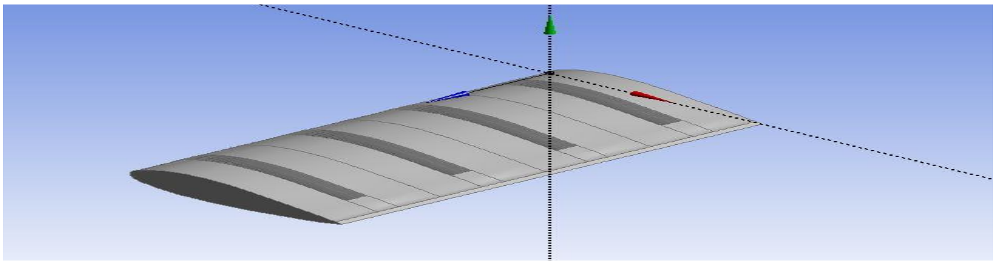


Figure 3: Wing with the Riblets.

4.1 Analysis

The Fluid Domain of the Wing model is shown below and the CFD analysis of the models With and without wing is carried out at 0, 2, Angle of Attack at 50 m/s. The details of the analysis conditions are explained in this chapter.

Domain-Default Domain	
Type	Fluid
Location	B1046
Materials	
Air Ideal Gas	
Fluid Definition	Material Library
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Not Buoyant
Domain Motion	Stationary
Reference Pressure	1.0000e+00[atm]
Heat Transfer Model	Isothermal
Fluid Temperature	2.0000e+01[c]
Turbulence Model	Laminar

5. Boundary Conditions

The analysis of the models is made at 50 m/s the boundary condition of the flow analysis is shown below,

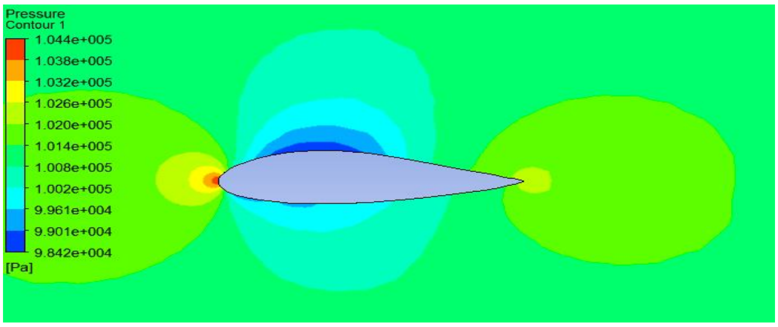
Boundaries	
Boundary-Inlet	
Type	Inlet
Location	F1051.1046
Settings	
Flow Regime	Subsonic

Mass And Momentum	Normal Speed
Normal Speed	5.0000e+01[ms^-1]
Boundary Outlet	
Type	Outlet
Location	F1049.1046
Settings	
Flow Regime	Subsonic
Mass And Momentum	Average Static Pressure
Pressure Profile Blend	5.0000e-02
Relative Pressure	1.0000e+00[atm]
Pressure Averaging	Average over whole outlet
Boundary- ATM	
Type	Wall
Location	F1047.1046, F1048.1046, F1050.1046, F1282.1046

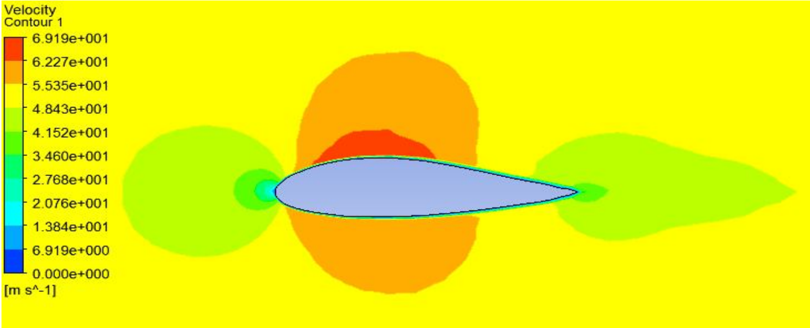
Figure: Boundary Details for 50 m/s.

6. Results

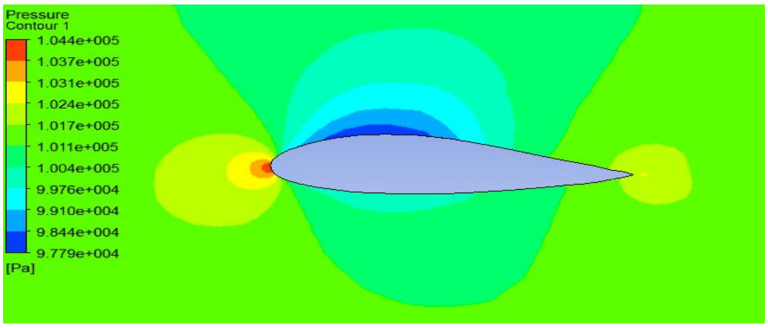
Pressure and velocity Plots without Riblets:



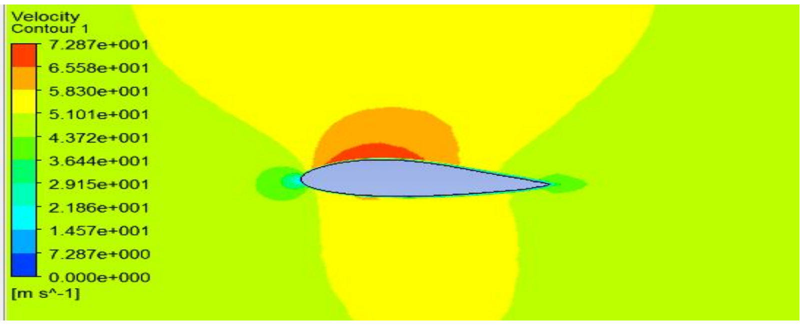
Pressure plot at 0 A0A – 50 m/s



Velocity plot at 0 A0A – 50 m/s

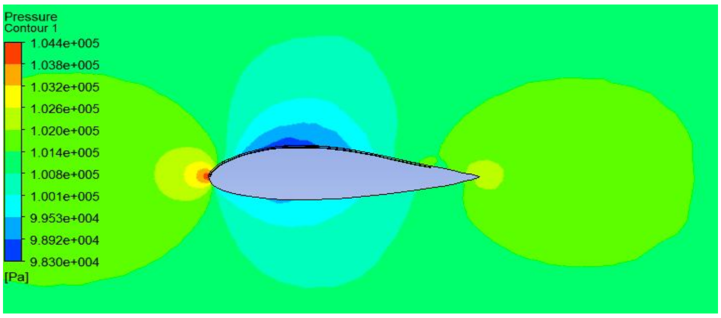


Pressure plot at 2 AOA – 50 m/s

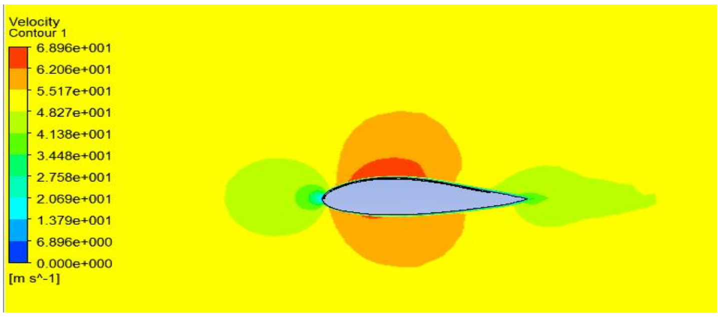


Velocity Plot at 2 AOA – 50 m/s

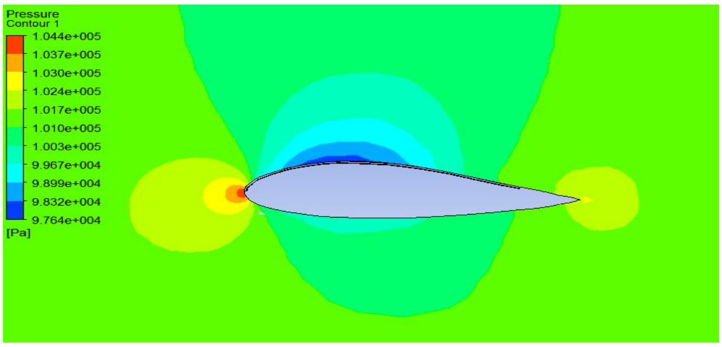
Pressure and Velocity Plots with Riblets:



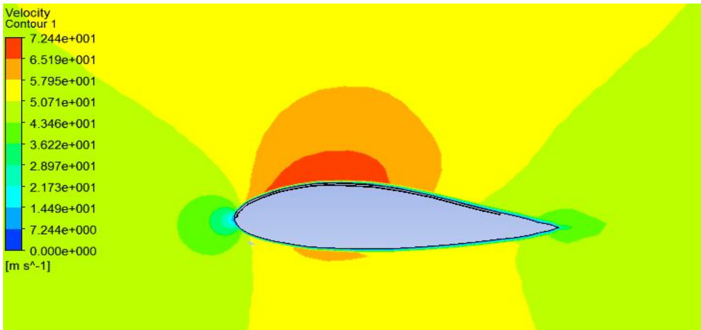
Pressure plot over Riblet at 0 AOA – 50 m/s



Velocity plot over Riblet at 0 AOA – 50 m/s



Pressure plot over Riblet at 2 AOA – 50 m/s



Velocity plot over Riblet at 2 AOA – 50 m/s

7. Result Comparison

Drag-50m/s	Wing without Riblet	Wing with Riblet	Reduction
0	1916.94	1748.42	168.52
2	1932.08	1791.81	140.27

8. Conclusion

The inference of the CFD investigation of flow over riblets in Drag reduction is that the riblets reduce the viscous drag of the wing at the low angle of attacks i.e. at the Angle of Attack below 5. After that there is increase in the drag value as the Angle of Attack increases. The Further Research of the Optimization of Riblets Shape can be carried out to further reduce the drag using Riblets.