International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 1, No. 3, October 2012

Research Paper

CFD DESIGN STUDY OF A CIRCULATION CONTROL INLET GUIDE VANE OF AN AEROFOIL

Manjunath Ichchangi[1]* and Manjunath H[1]

*Corresponding Author: **Manjunath Ichchangi,** ✉ manjumi08@gmail.com

Current projections for future aircraft concepts call for stringent requirements on high-lift and low cruise-drag. The purpose of this study is to examine the use of circulation control, through trailing edge blowing, to meet both requirements. This study is performed to validate of computational fluid dynamic procedures on a general aviation circulation control airfoil. In an effort to validate computational fluid dynamics procedures for calculating flows around circulation control airfoils, the commercial flow solver FLUENT was utilized to study the flow around a general aviation circulation control airfoil. The results were compared to experimental and computational fluid dynamics results conducted at the NASA Langley Research Center. This effort was performed and compared of the results for free-air conditions to those from previously conducted experiments.

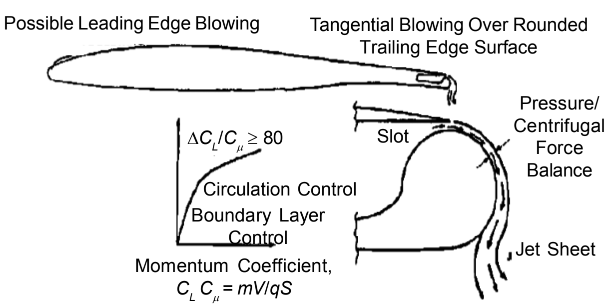
Keywords: GACC airfoil, Circulation control, Pressure, Velocity, Lift co-efficient

INTRODUCTION

The idea of the Circulation Control (CC) airfoil is by no means new; the concept has been around since the late 1930s. For this research, circulation control refers to changing the circulation of the airfoil using a stream of high-velocity air emanating from a slot near the trailing edge of the airfoil. Circulation control airfoils have historically been viewed as a means to obtain high lift. The majority of research efforts have focused on blowing in a positive, or downward, direction at the trailing edge of the airfoil. Early efforts accomplished this downward inclination using a jet of high-velocity air that is blown straight out of the trailing edge at the desired angle.This pneumatic-flap concept has been studied theoretically and experimentally by several researchers over the past several decades. As time has progressed, more researchers have begun to take advantage of the Coanda effect by blowing over a round trailing edge, as shown in Figure 1. This Coanda based circulation control is currently attracting

[1] Department of Mechanical Engineering, S.I.T, Tumkur 572103, India.

Figure 1: Basics of Circulation Control Aerodynamics



significant interest as a means of achieving high lift.

As is the case with all designs, there are trade-offs to be made for this increased performance. Issues such as mass-flow requirements and reduced efficiency due to trailing-edge bluntness when operating in conditions at which high lift is not needed have hindered the implementation of these circulation control airfoils on production aircraft. Typically these airfoils become undesirable when in cruise conditions due to the blunt trailing edge of many of the designs.

The Coanda effect occurs when the free stream flow above a curved surface is entrained by a parallel high momentum wall jet blown tangentially along the curved surface. The jet stays attached to the curved surface due to the balance between centrifugal forces around curved surface and the sub-ambient pressure in the jet sheet. The jet's momentum allows the oncoming boundary layer to overcome an adverse pressure gradient along the curved surface, and it entrains the flow above it due to its lower pressure. The entrained flow is accelerated around the curved surface by the jet, increasing the amount of circulation over the suction side of a body. This increased circulation translates to higher lift and flow turning for an airfoil that employs the Coanda effect. An example of the Coanda effect, applied to an inlet guide vane, can be seen in Figure 1, in which the flow is turned 11 degrees using a plenum pressure ratio of 1.8 (ratio of plenum pressure to inlet pressure).

THE CIRCULATION CONTROL WING CONCEPT

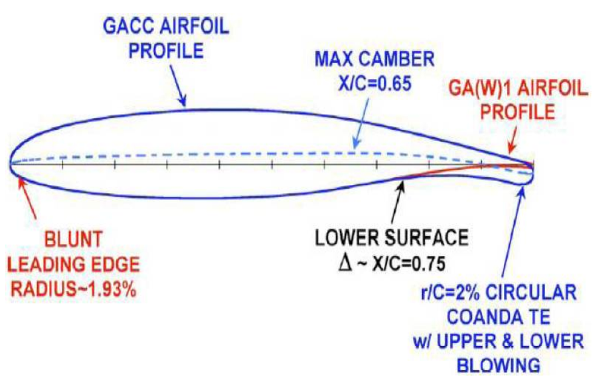
Conventional airfoils, such as the NACA series airfoils, all have a sharp trailing edge. The Kutta condition will be readily satisfied for this kind of the airfoil, and determines the circulation over the airfoil at a given free-stream condition and angle of attack. This sharp trailing edge design is very efficient for fixing circulation and lift, and is widely used both in nature and on man-made lifting surfaces. However, there are two limitations associated with it. First, the lift generated by a sharp trailing edge airfoil is only a function of angle of attack, camber, and free-stream conditions, and it can not be otherwise controlled. Secondly, the maximum lift achieved is limited, because the adverse pressure gradient on the upper surface eventually causes boundary layer separation and static stall with the increase in angle of attack. Thus, in order to obtain the high lift coefficient required during take-off or landing, high-lift devices must be used on a commercial aircraft. However, a high-lift system always contains many moving parts, and results in a significant weight penalty, and noise.

The Circulation Control (CC) airfoil overcomes these drawbacks in another way. It takes advantage of the Coanda effect by blowing a small, high-velocity jet over a highly curved surface, such as a rounded trailing edge. Since the airfoil trailing edge is not sharp, the Kutta condition is not fixed and the trailing edge stagnation point is free to move along the surface. In addition, the upper surface blowing near the trailing edge energizes the boundary layer, increasing its resistance to separation. With blowing, the trailing edge stagnation point location moves toward the lower airfoil surface, thus changing the circulation for the entire wing and increasing lift. Since the jet flow mass rate is readily controlled, this results in direct control of the separation point location, and thus the circulation and lift, as suggested by the name of this concept. Figure 2 shows a typical traditional CC airfoil with a rounded trailing edge.

GOVERNING EQUATIONS IN CFD

There are mainly three equations we solve in computational fluid dynamics problem. They are Continuity equation, Momentum equation (Navier Stokes equation) and Energy equation. The flow of most fluids may be analyzed mathematically by the use of two equations. The first, often referred to as the Continuity Equation, requires that the mass of fluid entering a fixed control volume either leaves that volume or accumulates within it. It is thus a "mass balance" requirement posed in mathematical form, and is a scalar equation. The other governing equation is the Momentum Equation, or Navier-Stokes Equation, and may be thought of as a "momentum balance".

Figure 2: Geometric Model



GRID GENERATED FOR FLUENT

Boundary Conditions

FLUENT does not allow the user to input freestream Mach number and Reynolds number directly (Figure 3). Instead, the free stream velocity and operating pressure were calculated using Equations (1)-(2) and provided as inputs for the analyses. The Mach

Figure 3: Grid Generated for Fluent

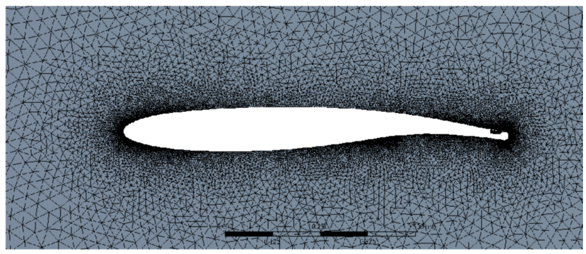
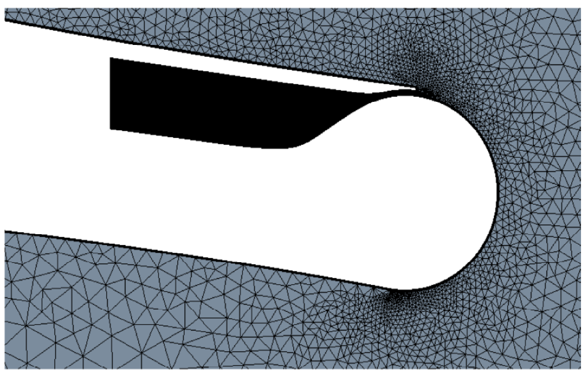
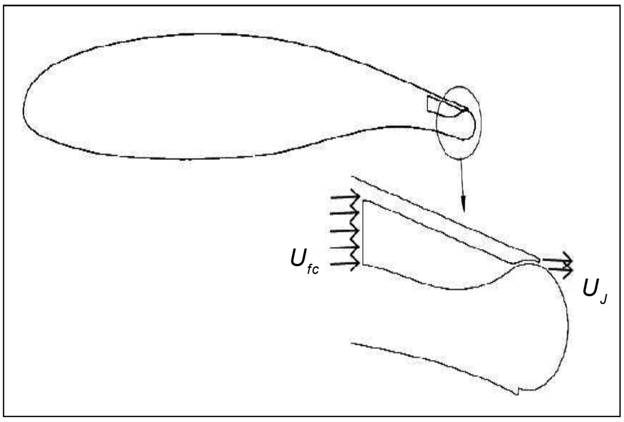


Figure 4: CFD Meshing Zoomed View-Close to CC Jet



and Reynolds numbers were set to 0.1 and 533,000, respectively, to match those used in according to Pugliese *et al.* (1979)



$$U_\infty = M_\infty \sqrt{\gamma R T_\infty} \quad \text{...(1)}$$

$$\rho_\infty = \frac{\mathrm{Re}\,\mu_\infty}{U_\infty c}$$

$$P_\infty = \rho_\infty R T_\infty \quad \text{...(2)}$$

An approximate method was developed to estimate the required velocity at the flow control boundary (U_{fc}) to achieve a desired $C\mu$, $C\mu$ desired. This method assumes incompressible flow throughout the duct, and was derived by solving the continuity equation. The equation for U_{fc} from this approximate method is given in Equation (3).

$$U_{fc} = U_\infty \sqrt{\frac{C_\mu A_J c b}{2 A_{fc}^2}} \quad \text{...(3)}$$

Results: $C\mu$ = 0.000

The CFD Results data shows that at positive angles of attack below approximately 5°, the flow remains laminar over the forward half of the airfoil. It then undergoes laminar separation followed by a turbulent reattachment (Figures 5 to 8).

Results with Circulation Control

Figure 5: Velocity Stream Lines at Various Angle of Sttack

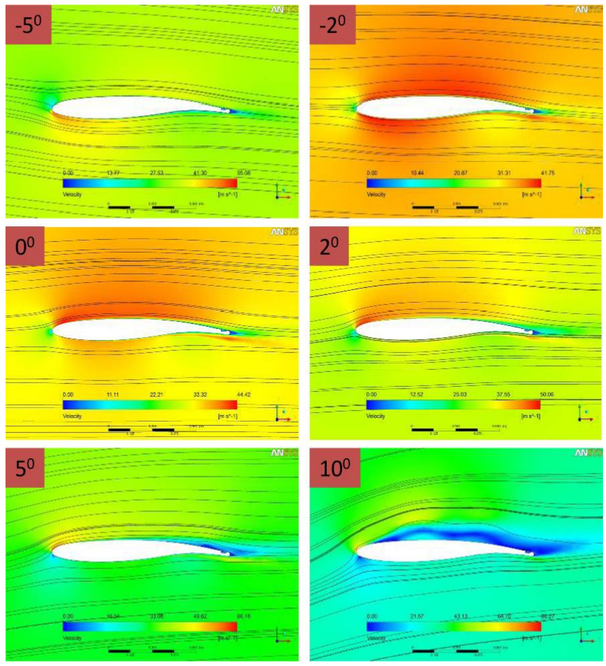


Figure 6: Velocity Contours of GACC Airfoil $C\mu$ = 0.015 for Fifferent AOA

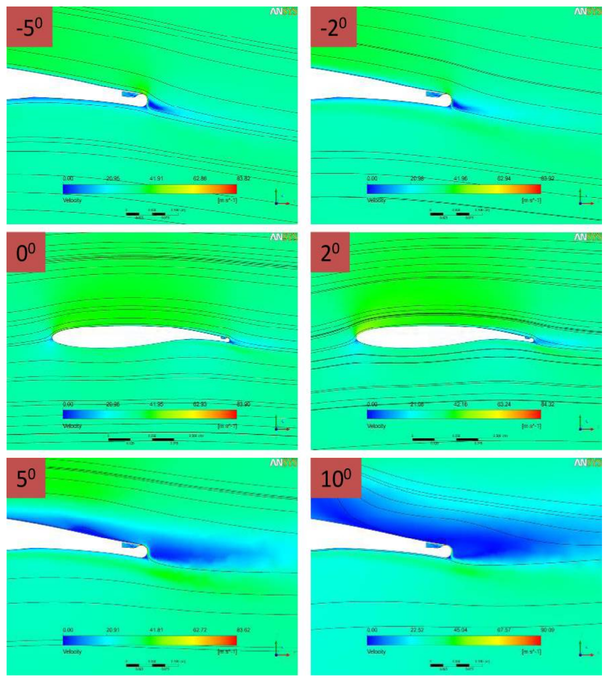


Figure 7: Velocity Streamlines of GACC Airfoil Different $C\mu$-AOA-5, 5 and 10°

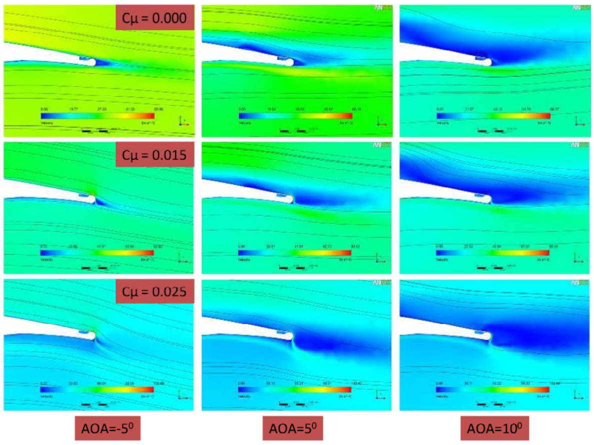
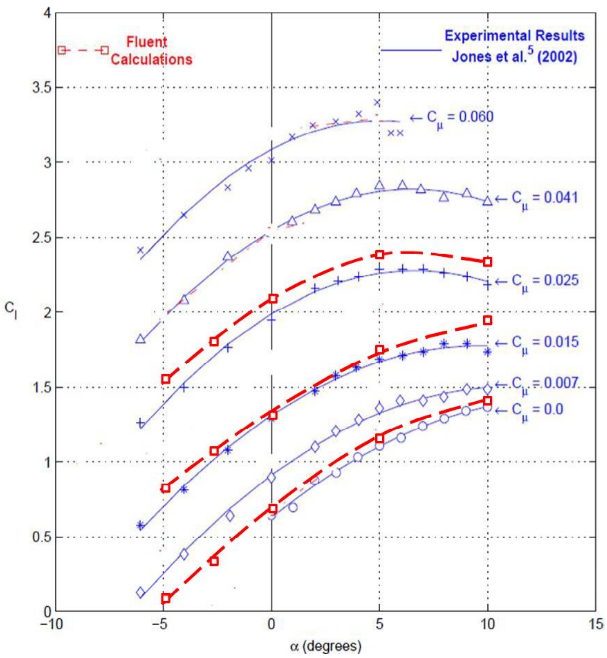


Figure 8: Comparison of Langley Experimental Results with the CFD Fluent Results



CONCLUSION

Current projections for future aircraft technologies call for challenging goals for both high lift for takeoff/landing conditions and low drag at cruise/climb conditions. Revolutionary approaches are needed to satisfy the demanding requirements. One approach is to explore the use of concepts that synergistically integrate aerodynamics and propulsion for achieving high efficiency at multiple operating conditions. The overall objective of this research effort is to explore the use of circulation control airfoils to achieve low drag at cruise and climb conditions while retaining the well-known very-high-lift capability of traditional circulation-control airfoils.

It can be seen that as the blowing rate is increased the streamlines become more curved—an indication of increased circulation. The the flow-field data the effects of changing the angle of attack while holding blowing rates constant. The results were presented for two blowing rates: the mild blowing case $C\mu$ = 0.015 and the highest blowing rate $C\mu$ = 0.025. The results shows that changes to $C\mu$ have a significant effect on the jet-separation location and the resulting C_1. In comparison, changes have a much smaller effect on the jet-separation location.

The values of $C\mu$ for the Fluent results match those for the results of Pugliese *et al.* (1979), it is clear that the trends and most of the predictions for the C_1 were close to those from Pugliese *et al.* (1979). In particular, the Fluent predictions for $C\mu$ = 0, 0.00, and 0.015 agree quite well with the results for similar values of $C\mu$ from Jones *et al.* (2002).

REFERENCES

1. Abramson J and Rogers E O (1983), "High-Speed Characteristics of Circulation Control Airfoils", AIAA Paper 1983-0265.

2. Baker W J and Paterson E G (2006), "RANS CFD Simulation of a Circulation-

Control Foil: Validation of Performance, Flow Field, and Wall Jet", AIAA Paper 2006-2010.

3. Bensor W A (1965), "Compressor Operation with One or More Blade Rows Stalled", pp. 341-364, NASA SP-36.

4. Budinger R E and Kaufman H R (1955), "Investigation of the Performance of a Turbojet Engine with Variable-Position Compressor Inlet Guide Vanes", NACA RME54L23a.

5. Constantin C Gheorghiu (Ed.) (1979), "Romanian Inventions and Prioritites in Aviation", Albatros Publishing House, Buchares.

6. Dobson W F and Wallner L E (1955), "Acceleration Characteristics of a Turbojet Engine with Variable-Position Inlet Guide Vanes", NACA RM-E54I30.

7. Englar R J and Huson G G (1983), "Development of Advanced Circulation Control Wing High Lift Airfoils", AIAA Paper 1983-1847.

8. Englar R J, Smith M J, Kelley S M and Rover R C (1993), "Development of Circulation Control Technology for Application to Advanced Subsonic Transport Aircraft", AIAA Paper 1993-0644.

9. Englar R J, Smith M J, Kelley S M and Rover R C (1994a), "Application of Circulation Control to Advanced Subsonic Aircraft, Part I: Airfoil Development", *Journal of Aircraft*, Vol. 31, No. 5, pp. 1160-1168.

10. Englar R J, Smith M J, Kelley S M and Rover R C (1994b), "Application of Circulation Control to Advanced Subsonic Aircraft, Part II: Transport Application", *Journal of Aircraft*, Vol. 31, No. 5, pp. 1169-1177.

11. Jones G S, Viken A E, Washburn L N, Jenkins L N and Cagle C M (2002), "An Active Flow Circulation Controlled Flap Concept for General Aviation Aircraft Applications", AIAA Paper 2002-3157.

12. Lord W K, MacMartin D G and Tillman T G (2000), "Flow Control Opportunities in Gas Turbine Engines", AIAA Paper 2000-2234.

13. Pugliese A J, Bethpage N Y and Englar R J (1979), "Flight Testing the Circulation Control Wing", AIAA Paper 1979-1791.

14. Viswanathan A K and Tafti D K (2004), "Numerical Analysis of Circulation Control on a NCCR 1510-7607N Airfoil Using RANS Models", 2004 Circulation Control Conference, March 16-17, Hampton, Virginia, USA. NASA/CP-2005-213509.

15. Wallner L E and Lubick R J (1955), "Steady State and Surge Characteristics of a Compressor Equipped with Variable Inlet Guide Vanes Operating in a Turbojet Engine", NACA RM-E54I28.

16. Wood N and Nielsen J (1985), "Circulation Control Airfoils Past, Present, Future", AIAA Paper 1985-0204.