IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 4 Ver. II (Jul- Aug. 2014), PP 64-71
www.iosrjournals.org

Numerical Simulation of Laminar Flow over Slotted Airfoil

P. B. Makwana[1], J. J. makadiya[2]
PG Student, Mechanical Engineering Department, RK University, Rajkot, Gujarat, Indi [1]
Assistant Professor, Mechanical Engineering Department, RK University, Rajkot, Gujarat, India[2]

Abstract: *In this paper mainly focused effect of fixed slot on performance of NACA0012 airfoil. Using slot lower surface high pressurized air passes through slot to energized upper surface. Analysis has been done on NACA0012 of 1m cord length at 25C of air with 5m/s air velocity. First plain airfoil at different angle of attack has been analyzed to find out stall condition and C_l, C_d. After that same parameter and physics, only change in geometry of airfoil with 15%C leading edge slot. Flow separation is adverse effect for performance of airfoil. Due to flow separation adverse pressure gradient effect reverse flow is there, we cause reduction of lift coefficient and increment of drag coefficient so flow separation necessary to reduce. There are different techniques to reduce flow separation but effect of slot on airfoil has been studied.*
Keyword: *Angle of attack (AOA), plain airfoil, slotted airfoil, SST model,*

I. Introduction

There are two common properties that need to be considered in order to evaluate the effectiveness of an airfoil: the lift-to-drag ratio and the maximum lift coefficient. These parameters, among many things, determine the stalling speed, potential pay-load and airplane maneuverability. CL max is determined by the airplane wing shape (e.g., airfoil, sweep)[1 to 3]. To enhance the aerodynamic performance, most recent designs utilize devices that further increase maximum lift coefficient. Such mechanical devices like apes and leading-edge slats are called high-lift devices. These devices are mostly used at take-o_ and landing; when the aircraft is at its minimum speed, and where delaying stall-speed is absolutely critical. Obviously, lift is much easier to generate as speed increases because of the higher dynamic pressures.

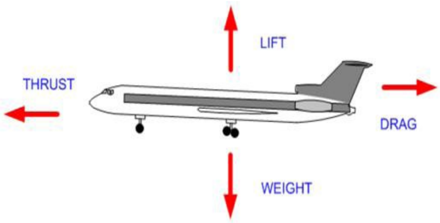


Figure 1 Forces acting on airplane [4]

A fixed-wing aircraft's wings, horizontal, and vertical stabilizers are built with airfoil-shaped cross sections, as are helicopter rotor blades. Airfoils are also found in propellers, fans, compressors and turbines. Sails are also airfoils, and the underwater surfaces of sailboats, such as the centerboard and keel, are similar in cross-section and operate on the same principles as airfoils. Swimming and flying creatures and even many plants and sessile organisms employ airfoils/hydrofoils: common examples being bird wings, the bodies of fish, and the shape of sand dollars. An airfoil-shaped wing can create down force on an automobile or other motor vehicle, improving traction. The airfoils are also characterized by their pressure and velocity distribution curves. The shape of the airfoil strongly affects the pressure distribution on the airfoil surface. By properly adjusting the airfoil shape it is possible to fine-tune the airfoil pressure distribution in order to adjust the airfoil performance. There is a great amount of airfoil shapes available in the literature and each one of these airfoils is characterized by its own performance curves. To select the proper airfoils for a aero plane wings design it is important to establish a set of boundary [5to10]

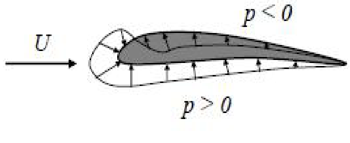


Figure 1.2 Pressure distribution on airfoil[4]

II. Governing Equation

Continuity Equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Momentum Equitation

$$\frac{\partial u}{\partial t} + \rho u\frac{\partial u}{\partial x} + \rho v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right]$$

$$\frac{\partial v}{\partial t} + \rho u\frac{\partial v}{\partial x} + \rho v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right]$$

III. Turbulence model

The Menter Shear Stress model is a two-layer model which employs the k-ω model of Wilcox (Ref. 24) in the inner region of boundary layers and switches to a k-ε model in the outer region of boundary layers and in mixing regions. The outer k-model is transformed to provide a second set of k-equations with a blending function used to transition between the two sets of equations. The SST model has been found to provide very good calculations of wall bounded flows even with highly separated regions. One example of this may be found in Ref. 25 where the SST model was found to provide the best predictions of several one- and two-equation models in the Wind code for separated nozzle flows. The details of the complete SST model are provided in Refs. 26 and 27, but here we only consider the outer equation set,.

IV. Result and discussion

4.1 Boundary condition

The NACA 0012 airfoil resides in a half elliptical computational domain whose upstream and downstream boundaries are located at 5 and 10 chord lengths from the leading edge, respectively. The upper and lower boundaries are placed at 5 chord lengths, each, from the leading edge. The no-slip condition is specified for the velocity on the airfoil surface while free-stream values are assigned for the velocity at the upstream boundary. At the downstream boundary a pressure outlet boundary condition is define. On the upper and lower surface boundaries the component of velocity normal to the component of stress vector along these boundaries is prescribed zero value. The Reynolds number based on the chord length of the airfoil, free-stream velocity and viscosity of the fluid is 10^5.

4.2 Finite element mesh

Fig. 1 shows a typical finite element mesh for plain airfoil is generated in Ansys while for slotted airfoil in Hypermesh for the computations. This mesh consists of 10875 nodes and 46946 tetra elements for slotted airfoil**.** The unstructured mesh provides flexibility to handle complex geometries. The structured mesh with Y^+ value as first layer thickness around the airfoil provides effective control on the grid to resolve the boundary layer.

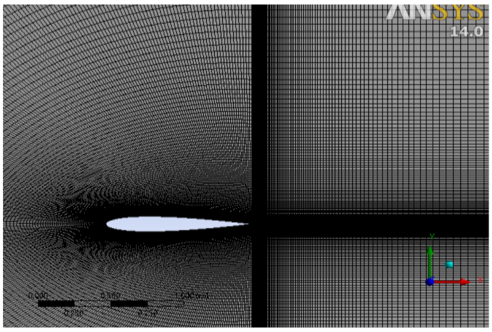


Figure 4.2.1 Meshing of plain airfoil

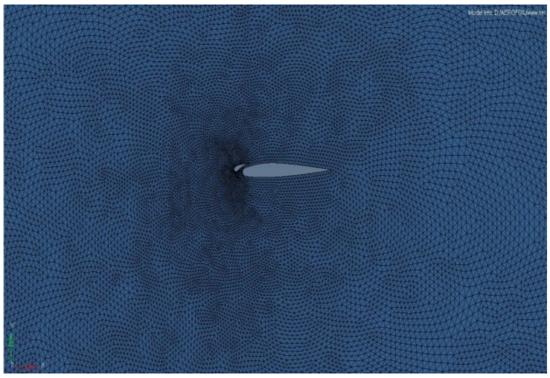


Figure 4.2.2 Meshing of slotted airfoil

V. Analysis at different angle of attack of plain airfoil

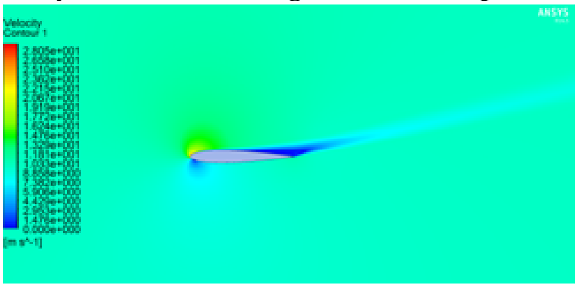


Figure 5.1 (a) Velocity Counter at 15 angle of attack

Figure 5.1(a) shows velocity counter at 15° angle of attack blue colour represent 0 m/s velocity, This is due to adverse pressure gradient effect flow separation occur. At strong adverse pressure gradient reverse

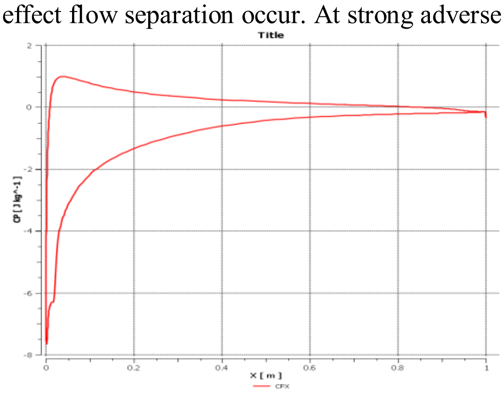


Figure 5.1(b) Cp Graph At 15 AOA

flow is there. Figure 5.1(b) shows Cp counter , it clearly indicate that reverse flow occur at 0.865%C(Cord length).

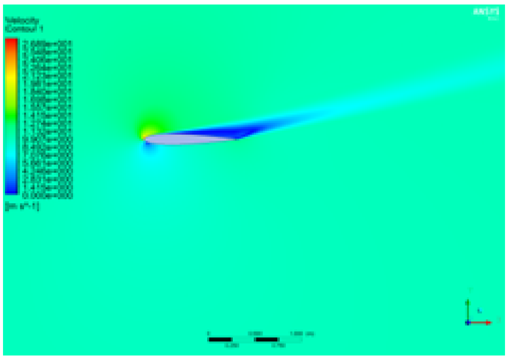


Figure 5.3 (a) Velocity counter at 16 angle of attack

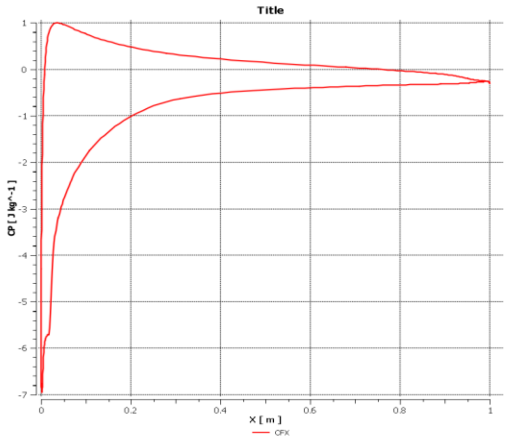


Figure 5.3(b) Cp graph at 16° AOA

Likewise figure 5.3 (a) and (b) represent velocity counter and Cp graph. From Cp graph it clearly shows that reverse flow occur at 0.8%C.

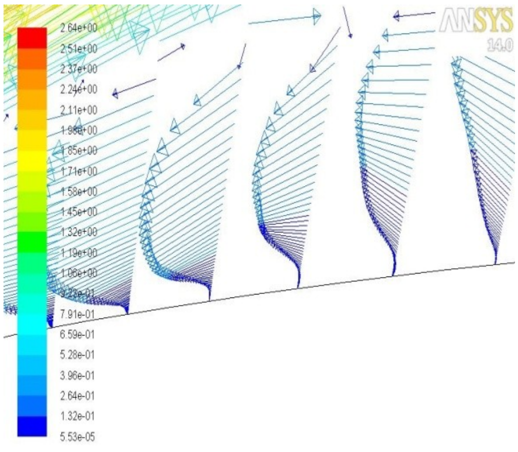


Figure 5.4 Velocity vector at 16° AOA

Figure 5.4 shows how velocity vector direction is reverse due to flow separation.

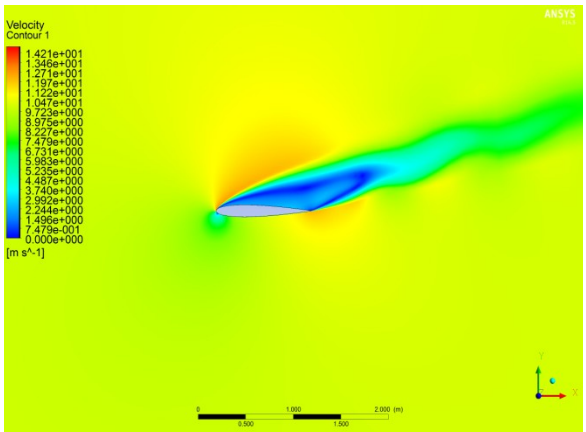


Figure 5.5(a) Velocity Counter at 17 angle of attack

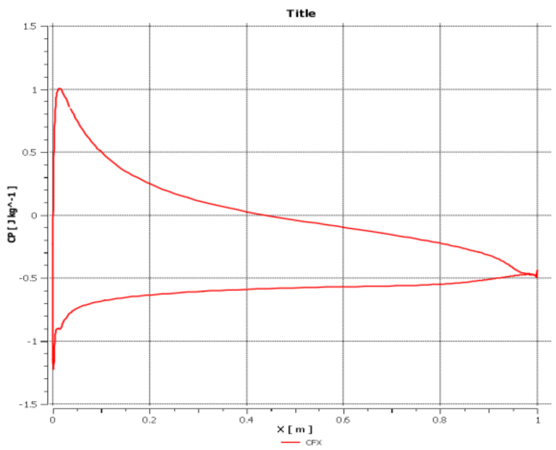


Figure 5.5(b) Cp graph at 17 angle of attack

Figure 5.5 (a) shows that flow separation occur from leading edge. From Cp graph it clearly shows reverse flow occur from 0.45%C.

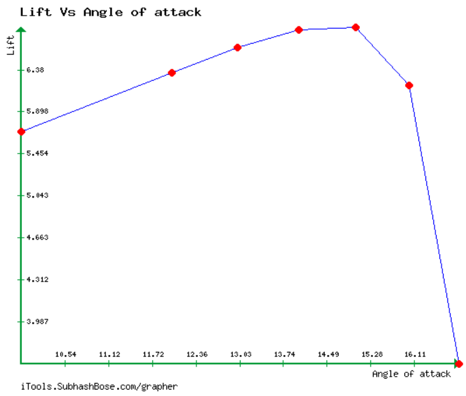


Figure 5.7 Graph of Lift force Vs. Different angle of attack

From previous result shows that with increasing angle of attack (AOA) lift force is going to increase and dreg force is going to decrease at certain AOA, after that lift force drastically going to decrease and dreg force is going to increase, so it's call stall condition . Figure 5.7 shows maximum lift occur at 15° AOA while stall condition occur at 17° AOA

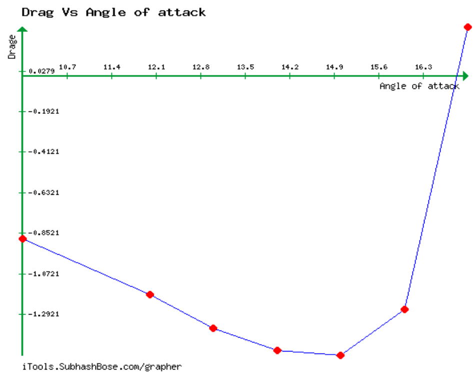
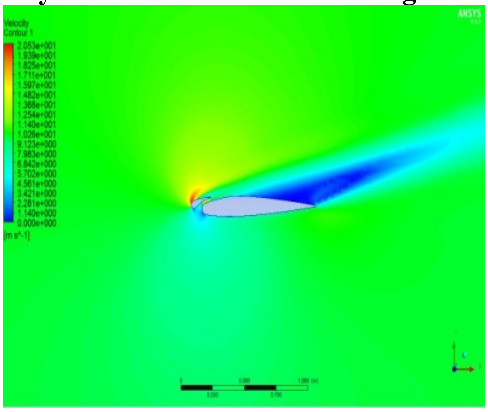


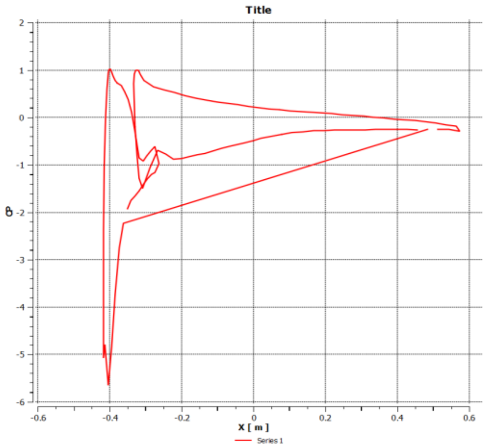
Figure 5.8 Graph of Dreg Vs. Different angle of attack

From figure 5.8 it clearly shows that minimum dreg occur at 15° and maximum at 17° AOA. To prevent flow separation there are different technique as mention in references [11-23]. Out of which by creating slot at leading edge to energized upper surface of airfoil by using high pressurized air from lower surface is effective.

VI. Analysis of Slotted airfoil at 17 angle of attack



(a)



(b)

Figure 6.1 (a) Velocity Counter of slotted airfoil **(b)** Cp graph at 17° AOA

Figure 6.1 (a) shows velocity counter of slotted airfoil at 17° AOA. Comparison of figure 5.5 and 6.1 shows that by using slotted airfoil flow separation is going to pull down wards trailing edge. Here it up to 0.9%C.

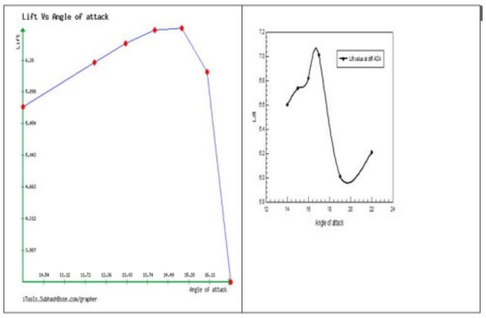


Figure 6.2 Comparison of plain and slotted airfoil for lift Vs. AOA

Figure 6.2 show that by using slotted airfoil stall condition is going towards higher angle of attack and also lift force is going to increase with slotted airfoil lift force is 7.12N while in plain it 7N

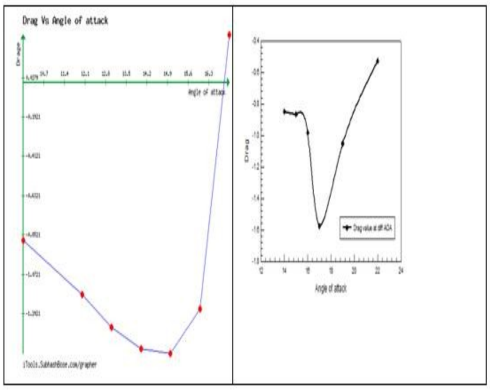


Figure 6.3 comparison of plain and slotted airfoil for dreg Vs. AOA

Figure6.3 shows that by using slotted airfoil dreg force is going to down with slotted airfoil minimum dreg occur at 17^{o} AOA while in plain airfoil minimum dreg at 15^{o}AOA.

VII. Conclusion

Analysis has been carried out in ANSYS 14.5 CFX by changing angle of attack with plain and slotted airfoil. By using leading slot upper layer of boundary is going to energize and try to pull down flow separation towards trailing edge.

1. As angle of attack is increasing flow separation is going towards leading edge.
2. For given parameter and physics stall condition for plain airfoil occur at 17o.
3. Maximum lift and minimum drag for plain airfoil occur at 15o.
4. With 15%C leading edge slot Maximum lift occur at 17o.
5. Slotted airfoil gives higher lift and lower drag then plain airfoil
6. Stall condition occur at higher angle of attack compare to plain

Acknowledgement

We would like to sincerely acknowledge the en-courageous efforts of Mechanical Engineering Department of R K School of engineering. Our heartfelt thanks to faculty members who helped us in prepare paper and give direction with their precious suggestions & rich experience.

Reference

[1]. John D Anderson Jr, Introduction to flight . McGraw-Hill Book Company, sixth ed.,2012.
[2]. E.L.Houghton, P.W.Carpenter. Aerodynamic for Engg. Student. CBS fourth ed. 2005
[3]. Yunus Cengel, Fluid mechanic . McGraw-Hill Book Company, sixth ed.,2009
[4]. F. M. White, Fluid Mechanics. McGraw-Hill, fifth ed., 2003.
[5]. E. Hau. Wind Turbines: Fundamentals, Technologies, Applications, Economics (2^{nd} Edition). Springer, 2006
[6]. Rong Ma, Peiqing Liu. "Numerical Simulation of Low-Reynolds-Number and High-Lift Airfoil S1223". Proceedings of the World Congress on Engineering 2009 Vol II
[7]. S.Kandwal , Dr. S. Singh. "Computational Fluid Dynamics Study Of Fluid Flow And Aerodynamic Forces On An Airfoil" International Journal of Engineering Research & Technology (IJERT), Vol. 1 Issue 7, September - 2012
[8]. Ji Yao, Jianliang Wang,Weibin Yuanb, Huimin Wang, Liang Cao. "Analysis on the influence of Turbulence model changes to aerodynamic performance of vertical axis wind turbine". Procedia Engineering 31 (2012) 274 – 281
[9]. Ji Yaoa, Weibin Yuan, jianliang Wang, Jianbin Xie , Haipeng Zhou , Mingjun Pengd, Yong Sun. "Numerical simulation of aerodynamic performance for two dimensional wind turbine airfoils". Procedia Engineering 31 (2012) 80 – 86
[10]. Evan Kontras, Kyle Gould, Davide Maffeo. " The NACA 4212 airfoil was analyzed using three separate methods". MAE 4440/7440 Aerodynamics University of Missouri
[11]. Hua Shan, Li Jiang, Chaoqun Liu "Direct numerical simulation of flow separation around a NACA 0012 airfoil" Computers & Fluids 34 (2005) 1096–1114
[12]. Sanjay Mittal*, Priyank Saxena "Hysteresis in flow past a NACA 0012 airfoil" Comput. Methods Appl. Mech. Engrg. 191 (2002) 2179–2189
[13]. M. Goodarzi, R. Fereidouni and M. Rahimi "Investigation of flow control over a NACA 0012 airfoil by suction effect on aerodynamic characteristics" Canadian Journal on Mechanical Sciences & Engineering Vol. 3 No. 3, June 2012
[14]. Apostolos Tentolouris Piperas, Martin O.L. Hansen. "Investigation of Boundary Layer Suction on a Wind Turbine Airfoil using CFD". Wind Energy Building 403 Kongens Lyngby Master's Thesis 5th August 2010
[15]. Armin Ghoddoussi. "A CONCEPTUAL STUDY OF AIRFOIL PERFORMANCE ENHANCEMENTS USING CFD". Submitted to Department of Aerospace Engineering and the faculty of the Graduate School of Wichita State University in partial fulfillment of the requirements for the degree of Master of Science(2011)
[16]. Deepanshu Srivastav "Flow Control over Airfoils using Different Shaped Dimples" IPCSIT vol.33(2012)©(2012) IACSIT Press, Singapore
[17]. Shutian Deng, Li Jiang, Chaoqun Liu. "DNS for flow separation control around an airfoil by pulsed jets". Computers & Fluids 36 (2007) 1040–1060

[18]. Kianoosh Yousefi, S.Reza Saleh & Peyman Zahedi. "Numerical Study of Flow Separation Control by Tangential and Perpendicular Blowing on the NACA 0012 Airfoil" International Journal of Engineering (IJE), Volume (7) : Issue (1) : 2013

[19]. Abbott, I. H., and von Doenhoff., A. E., Theory of Wing Sections, Dover, New York, 1959

[20]. S. M. Gottlieb, "Two-Dimensional Wind-Tunnel Investigation of Two NACA 6-Series Airfoils with Leading-Edge Slats," NACA RM No. L8K22, Langley Aeronautical Laboratory Langley Field, VA, Jan 1949.

[21]. JoonW. Lim. "Application of a Slotted Airfoil for UH-60A Helicopter Performance" NASA Ames Research Center Moffett Field, California (2002)

[22]. Hua Sha, Li Jiang, Chaoqun Liu, Michael Love, Brant Maines. "Numerical study of passive and active flow separation control over a NACA0012 airfoil" . Computers & Fluids 37 (2008) 975–992

[23]. D. K. Nowak and U. P. Solies, "Wind-Tunnel Tests of a High Lift Generation and Stall/Spin Recovery System," Journal of Aircraft, vol. 37, pp. 383-389, 2000

[24]. Wilcox, D.C., "Reassessment of the Scale-Determining Equation for Advanced

[25]. Turbulence Models," AIAA Journal, Vol. 26, No. 11, 1988, pp. 1299-1310

[26]. DalBello, T., Georgiadis, N.J., Yoder, D.A., and Keith, T.G., "Computational Study of Axisymmetric Off-Design Nozzle Flows," AIAA Paper 2004-0530, Jan. 2004

[27]. Menter, F.R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," AIAA Journal, Vol. 32, No. 8, Aug. 1994, pp. 1598-1605

[28]. Menter, F.R., "Zonal Two-Equation k-ε, ωTurbulence Models for Aerodynamic Flows," AIAA Paper 93-2906, July 1993