ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 7, July 2013

Assessment of boundary layer flow modeling approaches in computational fluid dynamics for compressible external aerodynamics using NACA-64618 Airfoil

Shankara Murthy.H.M[1], Madhukeshwara.N[2], S.Kumarappa[3]

PG Student (MTP), Department of Mechanical Engineering, BIET Davangere, Karnataka, India[1]

Assistant Professor, Department of Mechanical Engineering, BIET Davangere, Karnataka, India[2]

Professor and PG Coordinator, Department of Mechanical Engineering, BIET Davangere, Karnataka, India[3]

Abstract: In this work two dimensional CFD analysis have been carried out to predict the aerodynamic forces on the surface of the NACA 64618 airfoil at High Reynolds number of 9 million. The study focus on understanding the accuracy of the results obtained with different near wall modelling techniques. Computing the near wall flow region by two distinct approaches Viz. Direct resolution of shear stresses (Mentors SST Model) and use of wall function (K-ε Model) is assessed.CFD analysis results are compared with the wind tunnel test data available in the literatures. The reasons for deviation between CFD analysis results and wind tunnel test data are investigated and accurate procedure for modeling the near wall flow region is concluded.

Key Words: Computational Fluid Dynamics (CFD), Airfoil, Aerodynamic Coefficients, Lift, Drag, Turbulence Models, Stalling, Boundary layer, Turbulence model, Angle of attack.

I. INTRODUCTION

The rapid evolution of computational fluid dynamics (CFD) has been driven by the need for faster and more accurate methods for the calculations of flow fields around configurations of technical interest. In the past decade, CFD was the method of choice in the design of many aerospace, automotive and industrial components and processes in which fluid or gas flows play a major role. In the fluid dynamics, there are many commercial CFD packages available for modeling flow in or around objects. The computer simulations show features and details that are difficult, expensive or impossible to measure or visualize experimentally.

Viscous fluid flow analysis using Computational Fluid Dynamics (CFD) methods focuses mainly on accurate prediction of near wall boundary layer region. In the case of external aerodynamics flow, CFD analysis results like lift and drag forces depends greatly on the proper resolution of near wall physics, especially in the stalling region. Assessing the lift and drag forces on the sectional airfoil profile is a significant stage in Aerodynamic design of systems like Aircraft wings, Turbo machinery cascades, wind turbine rotor blade etc.

David Hartwanger et.al [06] conducted CFD analysis for NREL S809 airfoil section of the wind turbine blade using the 2d panel code X-Foil and ANSYS CFX 2D code. The work concluded that using a high-resolution structured mesh; with advanced turbulence and transition models provide an excellent match with experimental data in the attached flow regime. However, the CFD and XFOIL panel code over-predict peak lift and tend to underestimate stalled flow.

Franck bertagnolio et.al [07] discusses the experimental and 2D CFD simulation results for the NACA six digit wing section families. The work concluded that results obtained with Ellipsys 2D research code provides a good match with experimental data both in the attached and stalled flow regimes. Ellipsys 2D CFD code uses the shear stress transition

ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 7, July 2013

turbulence model to predict the turbulence effects accurately as well as the flow transition from laminar to turbulent regime. This option is not available in the commercial CFD solvers.

H.Gao et.al [08] conducted experimental and CFD simulations for corrugated dragonfly airfoil at low Reynolds number (Re=55000 to 68000).2D and 3D CFD simulations are conducted using MUSIC Navier-Stokes solver. The work concluded that 2D and 3D CFD results differ significantly at relatively high angle of attacks and 3D CFD results agree much better with the experimental data. It is concluded that vortex dominated flow at higher angle of attacks is strongly three-dimensional.

Literatures cited in this field indicates that the Turbulence models employed in most of the commercial CFD software's treat the boundary layer around the airfoil as fully turbulent and hence the drag forces are over predicted and the results obtained with these turbulence models underestimates the stalled region. Needs to be properly implemented in CFD analysis in order to have a reliable prediction in stalling, pre stall and post stall region.

II.MATHEMATICAL MODELS

The commercial FLUENT software package, FLUENT 6.3.26, was used for solving the set of governing equations. The numerical method employed is based on the finite volume approach. Fluent provides flexibility in choosing discretization schemes for each governing equation.

A.Governing Equations

The Governing Navier-Stokes equations for the flow physics considered in this work are written in vector form as

$$\frac{\partial U}{\partial t}+\frac{\partial G_1}{\partial x}+\frac{\partial G_2}{\partial y}+\frac{\partial G_3}{\partial z}=\frac{\partial G_{1v}}{\partial x}+\frac{\partial G_{2y}}{\partial y}+\frac{\partial G_{3v}}{\partial z}$$

Where,

U is a vector of conserved variables.

$$U=\begin{bmatrix}\rho\\ \rho u_1\\ \rho u_2\\ \rho u_3\\ \rho E\end{bmatrix}$$

G_1, G_2 and G_3 are the Inviscid Flux vectors.

$$G_1=\begin{bmatrix}\rho u_1\\ P+\rho {u_1}^2\\ \rho u_1u_2\\ \rho u_1u_3\\ Pu_1+\rho u_1E\end{bmatrix} G_2=\begin{bmatrix}\rho u_2\\ \rho u_2u_1\\ P+\rho {u_2}^2\\ \rho u_2u_3\\ Pu_2+\rho u_2E\end{bmatrix} G_3=\begin{bmatrix}\rho u_3\\ \rho u_3u_1\\ \rho u_3u_2\\ P+\rho {u_3}^2\\ Pu_3+\rho u_3E\end{bmatrix}$$

G_1v, G_2v and G_3v are the viscous flux vectors.

$$G_{1v}=\begin{bmatrix}0\\ \tau_{xx}\\ \tau_{xy}\\ \tau_{xz}\\ u_1\tau_{xx}+u_2\tau_{xy}+u_3\tau_{xz}-q_1\end{bmatrix} G_{2v}=\begin{bmatrix}0\\ \tau_{yx}\\ \tau_{yy}\\ \tau_{yz}\\ u_1\tau_{yx}+u_2\tau_{yy}+u_3\tau_{yz}-q_2\end{bmatrix} G_{3v}=\begin{bmatrix}0\\ \tau_{zx}\\ \tau_{zy}\\ \tau_{zz}\\ u_1\tau_{zx}+u_2\tau_{zy}+u_3\tau_{zz}-q_3\end{bmatrix}$$

τ_{xx},τ_{yy} and τ_{zz} are the normal stresses.

$$\tau_{xx}=2\mu\frac{\partial u_1}{\partial x}+\frac{2}{3}\mu\left(\frac{\partial u_1}{\partial x}+\frac{\partial u_2}{\partial y}+\frac{\partial u_3}{\partial z}\right)$$

$$\tau_{yy}=2\mu\frac{\partial u_2}{\partial y}+\frac{2}{3}\mu\left(\frac{\partial u_1}{\partial x}+\frac{\partial u_2}{\partial y}+\frac{\partial u_3}{\partial z}\right)$$

ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 7, July 2013

$$\tau_{zz} = 2\mu\frac{\partial u_3}{\partial z} + \frac{2}{3}\mu\left(\frac{\partial u_1}{\partial x} + \frac{\partial u_2}{\partial y} + \frac{\partial u_3}{\partial z}\right)$$

μ is the dynamic viscosity coefficient.
$(\tau_{xy} = \tau_{yx})$, $(\tau_{yz} = \tau_{zy})$, $(\tau_{xz} = \tau_{zx})$ Are the shear stresses given by

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial u_1}{\partial y} + \frac{\partial u_2}{\partial x}\right)$$

$$\tau_{yz} = \tau_{zy} = \mu\left(\frac{\partial u_3}{\partial y} + \frac{\partial u_2}{\partial z}\right)$$

$$\tau_{xz} = \tau_{zx} = \mu\left(\frac{\partial u_1}{\partial z} + \frac{\partial u_3}{\partial x}\right)$$

B.Turbulence models

The turbulence closure is taken by the K-ε Model and Mentors SST model

1) Wall function approach (k- ε model):

The transport equations for turbulent kinetic energy k and dissipation rate ε are written in tensor form as,

$$\left(\frac{\delta}{\delta xi}\right)(\rho ui\mathcal{E}) = \left(\frac{\delta}{\delta xi}\right)\left(\frac{\mu eff}{\sigma\mathcal{E}}\right)\left(\frac{\delta\mathcal{E}}{\delta xi}\right) + Pk - \rho\mathcal{E}$$

$$\left(\frac{\delta}{\delta xi}\right)(\rho ui\mathcal{E}) = \left(\frac{\delta}{\delta xi}\right)\left(\frac{\mu eff}{\sigma\mathcal{E}}\right)\left(\frac{\delta\mathcal{E}}{\delta xi}\right) + c\mathcal{E}1\left(\frac{\mathcal{E}}{k}\right)(P - c\mathcal{E}2\ \rho\mathcal{E})$$

The turbulence generation term P_k can be expressed in tensor notation as,

$P_k=\mu_t (\delta u_i/\delta x_i) [(\delta u_i/\delta x_j) +(\delta u_j/\delta x_i)]$

The turbulent viscosity is given by

$$,\mu_t = \rho c_\mu \frac{k}{\varepsilon}$$

The effective viscosity is given by,

$\mu_{eff} = \mu+\mu_t$

The near wall treatment to capture the viscous effects is taken care by standard wall functions.

2) Direct resolution of boundary layer region approaches (Mentors SST model):

The SST (Shear Stress Transport) model of Menter (1994) is an eddy-viscosity model which includes two main novelties:

1. It is combination of a k-ω model (in the inner boundary layer) and k-ε model (in the outer region of and outside of the boundary layer).
2. A limitation of the shear stress in adverse pressure gradient regions is introduced.

The k-ε model has two main weaknesses: it over-predicts the shear stress in adverse pressure gradient flows because of too large length scale (due to too low dissipation) and it requires near-wall modification (i.e. low-Re number damping functions/terms).

This model is almost identical to the Mentor baseline model. Only one constant (σ_{k1}) and the expression for turbulent eddy viscosity are different. The two-equation model (written in conservation form) is given by the following equation

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = P - \beta^*\rho\omega k + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_k\mu_t)\frac{\partial k}{\partial x_j}\right]$$

ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 7, July 2013

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho u_j\omega)}{\partial x_j}=\frac{\gamma}{\nu_t}P-\beta\rho\omega^2+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma_\omega\mu_t\right)\frac{\partial\omega}{\partial x_j}\right]\quad+2(1-F_1)\frac{\rho\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j}$$

Note that the Lagrangian derivative was used, which is not identical with the proper form of these equations as written by the author and others elsewhere. The equations have been written above to be in proper conservation form.

$$P=\tau_{ij}\frac{\partial u_i}{\partial x_j}$$

$$\tau_{ij}=\mu_t\left(2S_{ij}-\frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right)-\frac{2}{3}\rho k\delta_{ij}$$

$$S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$$

And the turbulent eddy viscosity is computed from

$$\mu_t=\frac{\rho a_1 k}{max(a_1\omega,\Omega F_2)}$$

C.Grid Independence Study

Structured grid is generated using ICEMCFD meshing software to discretize the computational domain. Grid size is varied from 15000 to 40000 elements for 2D CFD analysis .Mesh with element count of 40000 is found suitable from accuracy point of view. The mesh is optimized for standard wall function approach and the same mesh is modified near the airfoil wall as required by the Mentors Shear Stress transport turbulence model.

Sl No	Mesh size (No of elements)	CFD analysis results		y-plus
		C_l	C_d	
1	10000	0.423	0.0043	133
2	20000	0.521	0.0058	102
3	30000	0.591	0.0062	61
4	40000	0.612	0.0076	34

Table 1: Grid independence studies

Experimental values of C_l= 0.62, C_d= 0.008 for an angle of attack of 6 degrees.

D.Boundary conditions

There are three faces bounding the calculation domain namely: the inlet boundary, the wall boundary and the outlet boundary. No-Slip boundary conditions are applied along the solid walls and wall functions were used as described earlier.The inlet boundary conditions involve total pressure and velocity direction cosines for varying angle of attack, turbulence intensity and turbulent viscosity ratio. The velocity components are calculated for each angle of attack as follows. The x-component velocity direction cosine is calculated by $x=ucos\alpha$ and the y component velocity direction cosine is calculated by $y=usin\alpha$, where α is the angle of attack in degrees. The inlet velocity (U) is 132 m/sec for a free stream Reynolds number of 9 million and air at STP (Temperature=15^{0}c, Pressure=1.01325 bar) as the fluid medium. Ambient

ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 7, July 2013

atmospheric condition is imposed at outlet. Turbulent intensity and viscosity ratio conditions are set to a value of 5 as per the industry practices.

E.Geometrical Modeling and Grid Generation

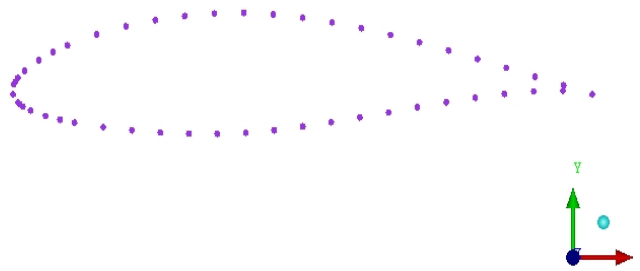

Fig 1: NACA-64618 Airfoil Coordinates

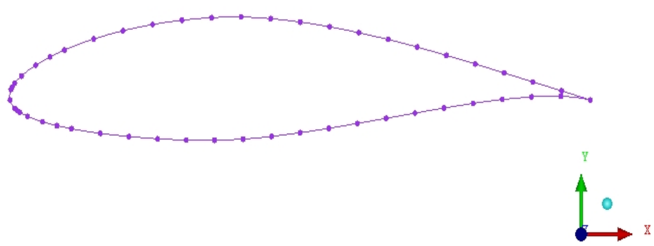

Fig 2: NACA-64618 Airfoil

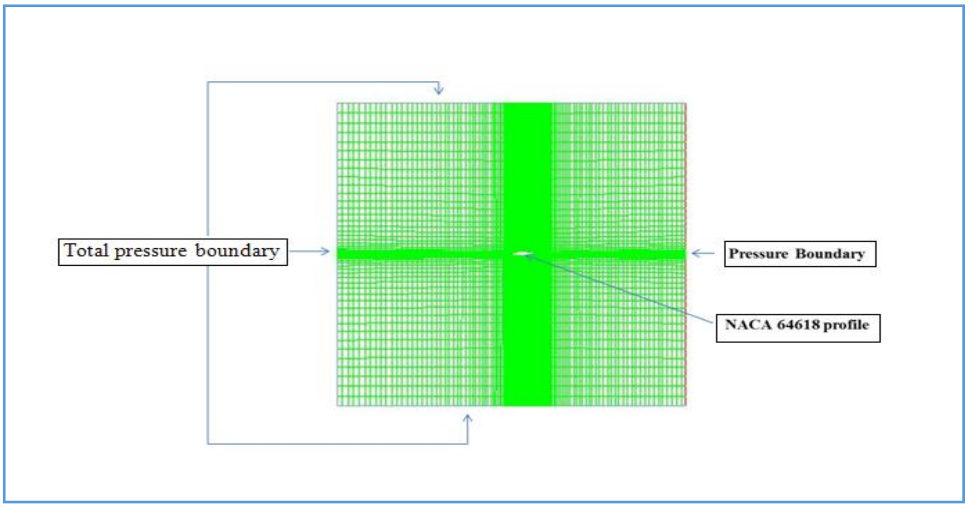

Fig 3: Geometry for 2D CFD analysis

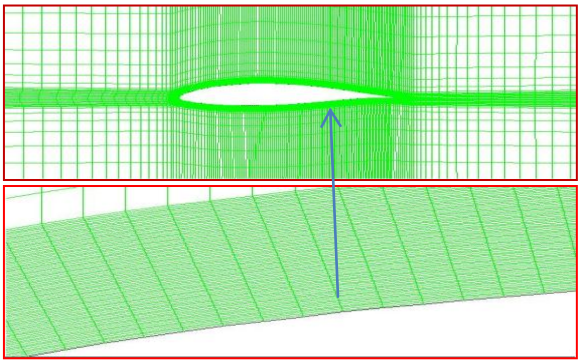

Fig 4: Grid resolution around the airfoil for
K-epsilon model(wall function approach)

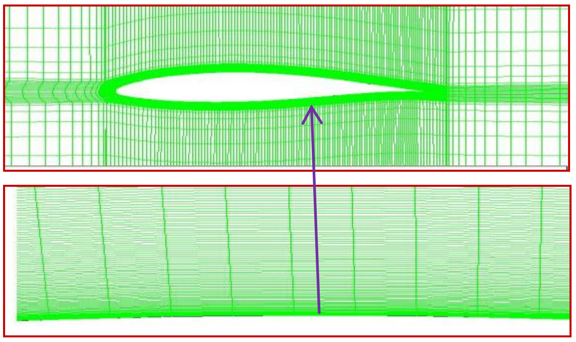

Fig 5: Grid resolution around the airfoil for
Mentors Shear stress transport turbulence model

ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 7, July 2013

III.RESULTS AND DISCUSSIONS

Table 2: Lift and Drag coefficients obtained by different models

AOA	Wind tunnel test results (Experimental)		K-epsilon model		Mentors SST model	
	C_l	C_d	C_l	C_d	C_l	C_d
0	0.4	0.0048	0.39	0.0044	0.395	0.0045
2	0.6	0.005	0.58	0.0045	0.59	0.0048
4	0.9	0.0053	0.891	0.0047	0.895	0.0051
6	1.1	0.0086	1.01	0.0075	1.05	0.0082
8	1.28	0.0132	1.18	0.0110	1.2	0.013
10	1.43	0.0148	1.32	0.0137	1.41	0.0145
12	1.5	0.015	1.39	0.0141	1.48	0.0148
14	1.55	0.0165	1.46	0.0153	1.5	0.0161
16 (Stalling)	**1.6**	**0.017**	**1.48**	**0.0159**	**1.59**	**0.0165**
18	1.58	0.018	1.45	0.0165	1.57	0.017
20	1.56	0.02	1.43	0.0174	1.54	0.0185

Table 2 shows the lift and drag coefficients obtained by the wind tunnel test (experimental) results, they are taken from the books "theory of wing sections" by Abott. And this table also contains the lift and drag coefficients obtained by the two different modeling approaches that is K-ε model and Mentors SST model, comparing these approaches with the wind tunnel results.

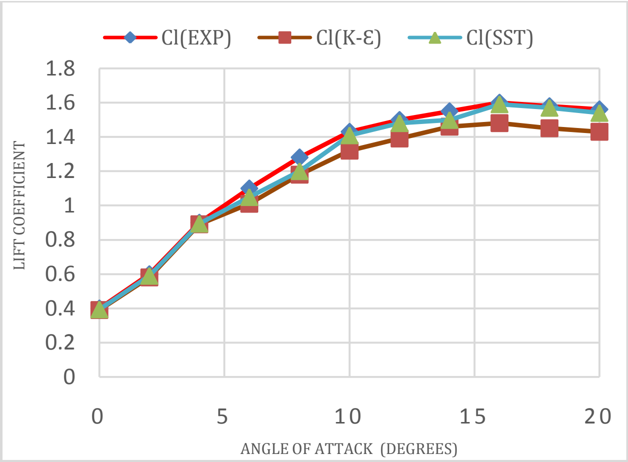

Fig6: Lift coefficient v/s AOA

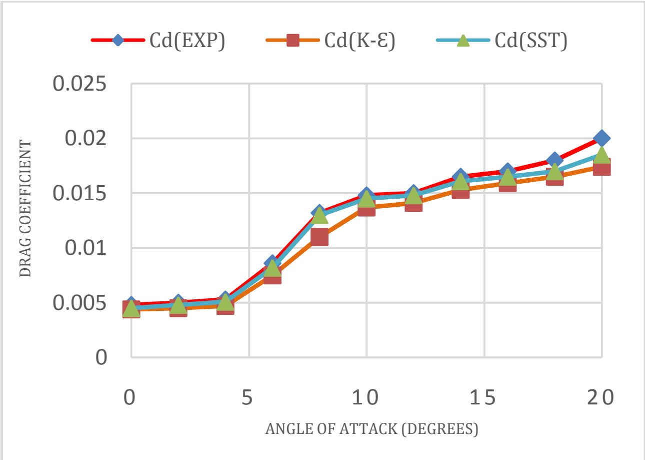

Fig7: Drag coefficient v/s AOA

ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 7, July 2013

From the wind tunnel test data NACA-64618 subsonic Airfoil gives a maximum lift coefficient of 1.6 at an angle of 16 degree for Reynolds No of 9 million. The Stalling starts from angle of attack 12 degree and there is only a slight increase in lift coefficient value from angle of attack 12 to 16 degrees and is shown in figure6.

Figure6 and 7 shows the lift and drag coefficient computed with K-epsilon turbulence model with standard wall function and Mentors shear stress turbulence model. It is observed that lift and drag coefficients computed with Mentor's shear stress model are close to the experimental results measured in the wind tunnel.

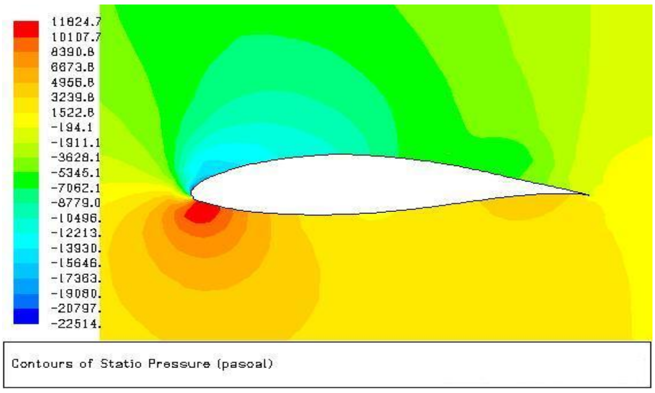

Fig 8: Pressure vector at 14^0 AOA (Pre-Stall) by SST Model

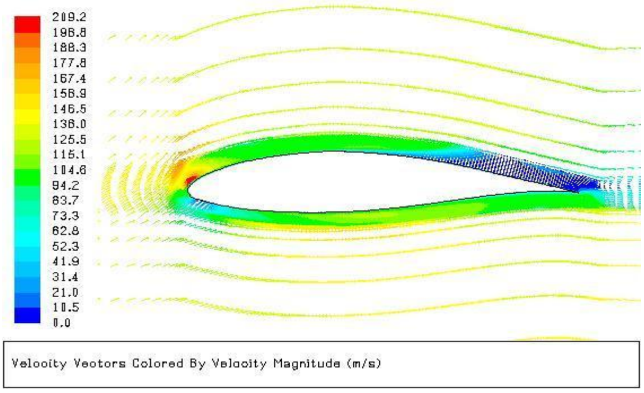

Fig 9: Velocity vector at 14^0 AOA (Pre-Stall) by SSTModel

The pre-stall region is the region occurs before the stalling point, from the velocity and pressure vectors shown in Fig 8 and 9, concluded that it is the initiative point for stalling, means that reducing in the velocity at the upper surface of the trailing edge of the airfoil. This results in reducing the lift and increasing in drag force.

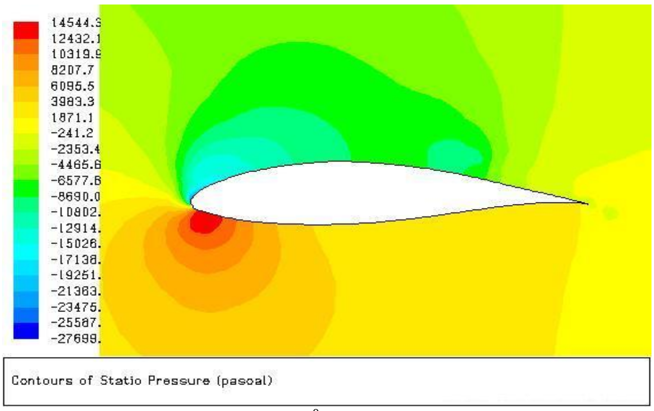

Fig 10: Pressure vector at 16^0 AOA (Stalling) by SST Model

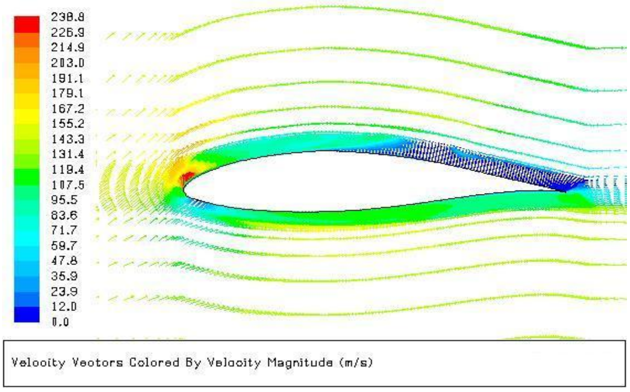

Fig 11: Velocity vector at 16^0 AOA (Stalling) by SST Model

Stalling point is the point at which the reduction in the lift force is maximum and increasing in Drag, and also the velocity at the upper surface of the trailing edge is almost equal to zero that we see from figures 10 and 11.

It is observed that the flow separation region is well captured by the Mentors shear stress model and hence the Mentors shear stress model is correctly predicting the lift coefficient in the stalled region.

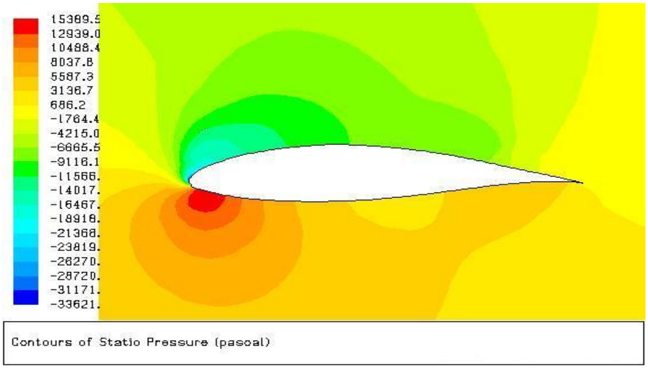

Fig 12: Pressure vector at 18^0 AOA (Post-Stall) by SST Model

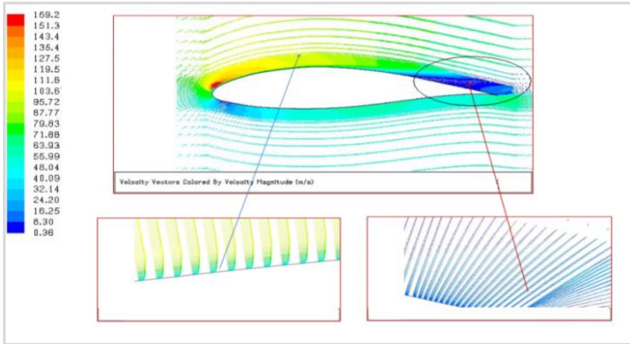

Fig 13: Velocity vector at 18^0 AOA (Post-Stall) by Mentors SST turbulence model

From figure 13 it is observed that the flow separation region is well captured by the Mentors shear stress model and hence the Mentors shear stress model is correctly predicting the lift coefficient in the stalled region.

IV. CONCLUSION

From the above discussions the lift and drag coefficients taken from the experimental wind tunnel test results are 1.6 and 0.017 respectively at an angle of attack 16^0 (Stalling Region).similarly 1.48 and 0.0159 are Lift and Drag coefficients obtained by the K-ε Model, 1.59 and 0.0165 are Lift and Drag Coefficients obtained by the Mentors SST turbulence model.

It is observed that lift and drag coefficients computed with Mentor's shear stress model are close to the experimental results measured at the wind tunnel. And the flow separation region is well captured by the Mentors shear stress model and hence the Mentors shear stress model is correctly predicting the lift coefficient in the stalled region.

Hence it is concluded that Mentors shear stress transport Turbulence model employing the direct resolution of boundary layer region by a fine mesh gives the accurate CFD analysis results in the pre stall as well as post stall region.

REFERENCES

[01] Abbott.I.H,'Theory of wing section, including a summary of airfoil data", Dover book on Physics, 1995

[02]Anderson DA., Tannehill JC. andPletcher, RH. "Computational Fluid Mechanics and Heat Transfer, " Hemisphere Publishing Corporation, McGraw-Hill Book Company, 1984

[03] Anderson John D,"Computational Fluid Dynamics", McGraw-Hill, 1995.

[04] John Anderson,Jr,"Introduction to Flight", McGraw –Hill, 2000

[05] F.M.White,"Fluid Mechanics", McGraw-Hill, 2005

[06] David Hartwanger et.al "3 D modeling of a Wind Turbine using CFD"NAFEMS Conference, United Kingdom, 2008

[07]FrankBertagnolio et.al "Wind Turbine airfoil catalogue"RISOE National Laboratories, Denmark, 2001

[08] H.Gao et.al "Computational study of unsteady flows around dragonfly and smooth airfoils at low Reynolds number"46th AIAAAerospace sciences meeting and exhibit,Reno,
Navada, 2008

[09] Vance Dippold, III," Investigation of Wall Function and Turbulence Model Performance within the Wind Code", 43rd AIAA Aerospace Sciences Meeting and Exhibit, 10 - 13 January 2005, Reno, Nevada

[10]J.E.Bardiana, et.al "Turbulence modeling validation study"
NASA Technical Memorandum 110446, 1997

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

[12] D.C.Wilcox,'Turbulence modelling for CFD'DCW Industries, California, 1993.

ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 6, June 2013