SPALART ALLMARAS UNSTEADY FLOW INVESTIGATION USING COMPUTATIONAL FLUID DYNAMICS

N. Verma[1] , Dr. S. Singh[2]

[1]M. Tech Scholar, [2]Associate Professor

Department of Mechanical Engineering,
Bipin Tripathi Kumaon Institute of Technology, Dwarahat, Almora, Uttarakhand (India) 263653

Abstract

This paper presents Spalart Allmaras flow investigation. In this paper, a computational fluid dynamics CFD model is studied for unsteady external flow. The model is designed in GAMBIT 2.3.16 and implemented with the help of FLUENT 6.3.26. Dynamic pressure and velocity magnitude are analysed. Fluid medium is taken atmospheric air for the analysis. Analytical solution of the governing equation of Spalart Allmaras flow is also illustrated.

Keywords: *FLUENT 6.3.26, unsteady flow, Grid, Boundary layer, GAMBIT 2.3.*

1. Introduction

Spalart Allmaras (SA) is a turbulence model for modeling different type of turbulent flows, specifically aerodynamic flows. Different terms of the governing equation of this model (e.g. production, diffusion and destruction), were modified by Spalart Allmaras after 1992.

The external flow steady is an important for the designing industry and in over daily operation also. Turbulent wall jet are widely used in many engineering process such as inlet devices in ventilation, separation control in airfoils and film cooling of turbine blades.

2. Mechanism of Internal flow

In fluid mechanics, **external flow** is such a flow that boundary layers develop freely, without constraints imposed by adjacent surfaces. Accordingly, there will always exist a region of the flow outside the boundary layer in which velocity, temperature, and/or concentration gradients are negligible. It can be defined as the flow of a fluid around a body that is completely submerged in it. Turbulence is flow characterized by recirculation, eddies, and apparent randomness. It is believed that turbulent flows can be described well through the use of the Navier–Stokes equations. Direct numerical simulation (DNS), based on the Navier–Stokes equations, and makes it possible to simulate turbulent flows at moderate Reynolds numbers. Turbulent flows are unsteady by definition. A turbulent flow can, however, be statistically stationary. According to Pope the random field *U*(*x*, *t*) is statistically stationary if all statistics are invariant under a shift in time.

3. Literature review

The history of turbulence modeling can first be traced back to Leonardo da Vinci's drawing from the fifteenth century. Later work by Newton, Euler, Bernoulli, d' Alembert, Navier, Fourier, B. de St. Venant and Stokes led to a viscous fluid model with thermal conduction. With past century, work by Reynolds, Prandtl, Von karman and Taylor finally led to a mathematical model for turbulent fluid motion based on assumption of continuum flow, averaged flow, viscous flow and the obedience to a set of turbulent postulates.

A large number of research analyses have been carried out on the external flows during the recent years. **Spalart Allmaras (1992)** mathematically modelled the single equation model used basically for aerodynamics. John Gatsis also investigate the Spalart Allmaras model.

4. Analytical Solution

SPALART ALLMARAS is a turbulent model having single equation, which is basically used for aerodynamic flows. There are various term for the Reynolds stress in the equation, which are identified as convection, diffusion, production, destruction. In the equation viscosity is a dependent variable, which is directly related to the Reynolds stress.

$$v_t = \frac{\overline{u'v'}}{du/dy}$$

According to convection low

$$\frac{DF}{Dt} = \frac{\partial F}{\partial t} + (u.\nabla)F$$
$$= Diffusion + production - Destruction$$

General definition of diffusion is.

$$Diffusion = \nabla.\varphi_F$$

Here φ_F is flux of scalar F due to the diffusion

$$\varphi_F = D_F \nabla F$$

Here D_F is coefficient of diffusion.

Total viscosity $n = \in + \gamma$

Where n is eddy viscosity, γ is molecular viscosity.

Nee & Kovasznay assumed that due to turbulent motion the coefficient of diffusion $D_n = n$, so the Schmidt number are approximetly equal to the $D_n/n \approx 1$.

$$Diffusion = \nabla.(n\nabla n)$$

SA represent the diffusion term quite differ to the NK model as.

$$\nabla.([{}^{v_t}/_{\sigma}]\nabla v_t),$$

Here v_t is eddy viscosity and σ is turbulent Prandtl number.

$$Diffusion = \frac{1}{\sigma}[\nabla.(v_t \nabla v_t) + c_{b2}(\nabla \mathrm{v_t})^2]$$

For the production term, in the SA flow they used the magnitude of vorticity w

$$w = \sqrt{2\Omega_{ij}\Omega_{ij}}$$

Where $\Omega ij = \partial U_i/\partial x_j - \partial U_j/\partial x_i$

$$Productioon = c_{b1}S_{vt}$$

$$s \equiv \sqrt{2\Omega_{ij}\Omega_{ij}}$$

$$\Omega_{ij} = \frac{1}{2}(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i})$$

Destruction eddy viscosity is the reason of turbulent flow is the assumption used here for destruction by NK.NK says that decay of energy is inversely proportional to the square of the energy as follows:

$$\frac{\overline{du}^2}{dt} = -\beta\overline{(u)}^{2\,2}$$

By integrating

$$\overline{u^2} \cong 1/\beta t$$

Consider the quantity F to be the total viscosity n.

$$\frac{\partial n}{\partial t} = -Destruction$$

By NK $Destructon \cong \beta n^2$ based on assumption and dimensional aspect the final form of destruction is as follows

$$Destruction = \frac{B}{L^2}\, n(n - v)$$

L is characteristic length and B is non dimensional universal constant, L is a function of y and considered to be equal to δ, δ is the boundary layer thickness closer to wall, L=Y.

In SA destruction term is closer to NK representation. The difference is that SA defined a non dimensional function besides a constant one. First form of destruction would be $-c_{w1}\left(\frac{v_t}{d}\right)^2$

Where d is the distance to wall and c_{w1} is constant. Due to the distance term it forms a log layer. On the other side it produces a low skin friction coefficient.

SA multiplies the first form of destruction by non dimensional function f_w. This is equal to 1 in log layer.

$$Destruction = -c_{w1}f_w(\frac{v_t}{d})^2$$

Final Form of Spalart Allmaras Model

SA introduces two new variables for buffer and viscous sub layer. $\bar{u}$ This is equal to ut.

SA does not require a fine grid as compare to K-w or K-∈ model. In order to arrive at SA, considered the classical log layer and devise near wall. These functions are different than $f_{w.}$

New variable $u = Kyu_T$ in the log layer but not in buffer layer

This result to the following equation

$$u_t = \bar{u} * f_{u1},$$

$$f_{u1} = \frac{X^3}{X^3 + c^3{}_{u1}}$$

Definition of scalar norms of strain rate tensor S is

$$\tilde{S} \equiv S + \frac{\bar{u}}{k^2 d^2} f_{u2},$$

$$f_{u2} = 1 + \frac{X}{1 + X f_{u2}}$$

Where f_{u2} has constructed just like f_{u1}.

Final form of the basic governing equation of Spalart Allmaras model is as follows

$$\frac{D_{\bar{u}}}{D_t} = c_{b1}\bar{S}\bar{u} + \frac{1}{\sigma}\left[\nabla.\left((v + \bar{u})\nabla\bar{u}\right) + c_{b2}(\nabla\bar{u})^2\right] - c_{w1} f_w \left[\frac{\bar{u}}{d}\right]^2$$

5. Modelling and Simulation

The whole designing work is carried out with the help of ‘GAMBIT 2.3.16’ and the analysis work is carried out in ‘FLUENT 6.3.26’. FLUENT 6.3.26 is computational fluid dynamics (CFD) software package to stimulate any kind of fluid flow problems. It uses the finite volume method to solve the governing equations for a fluid.

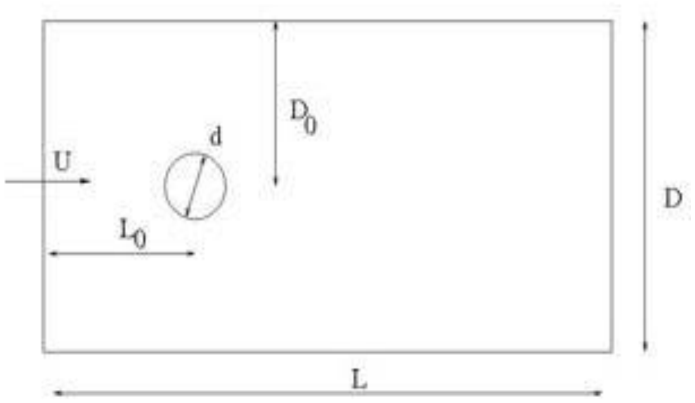
While Geometry and grid generation is done using GAMBIT 2.3.16 which is the pre-processor bundled with FLUENT.

6. Results and Discussion

For Turbulence flow, especially the Spalart Allmaras flow the analysis gives the appropriate result. In figure 2 the drag convergence is being constant after 60 time step, about 2700 iteration. In the figure 1 the turbulent seen nearby the object is higher and decreases as the distance increase. The turbulence is higher at the initial time, this is shown in the figure 3.

The net moment calculated by CFD is 15.4987 nm, where viscous moment is 1.614349 nm rest of moment is pressure moment. The net force is 102.285N, where pressure force is 82.491917N and viscous force is 19.793083N.

S.No.	Parameter	Value
1	Diameter	2m
2	Flowing Fluid	Air
3	Velocity of Air	1m/s
4	Reynolds Number	150
5	Model	SA
6	Density	75kg/m3
7	Temperature	288.16k
8	Specific Heat	1.4



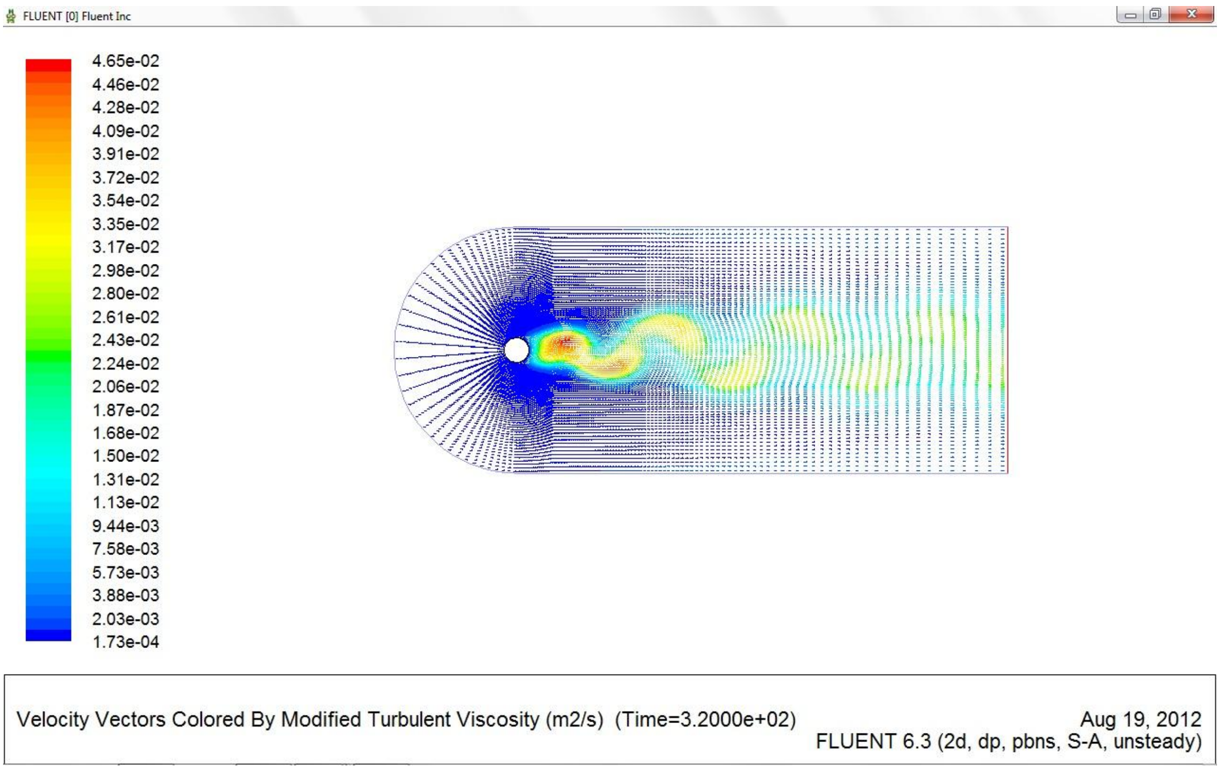


Figure 1: Velocity Vector by Modified Turbulent Viscosity

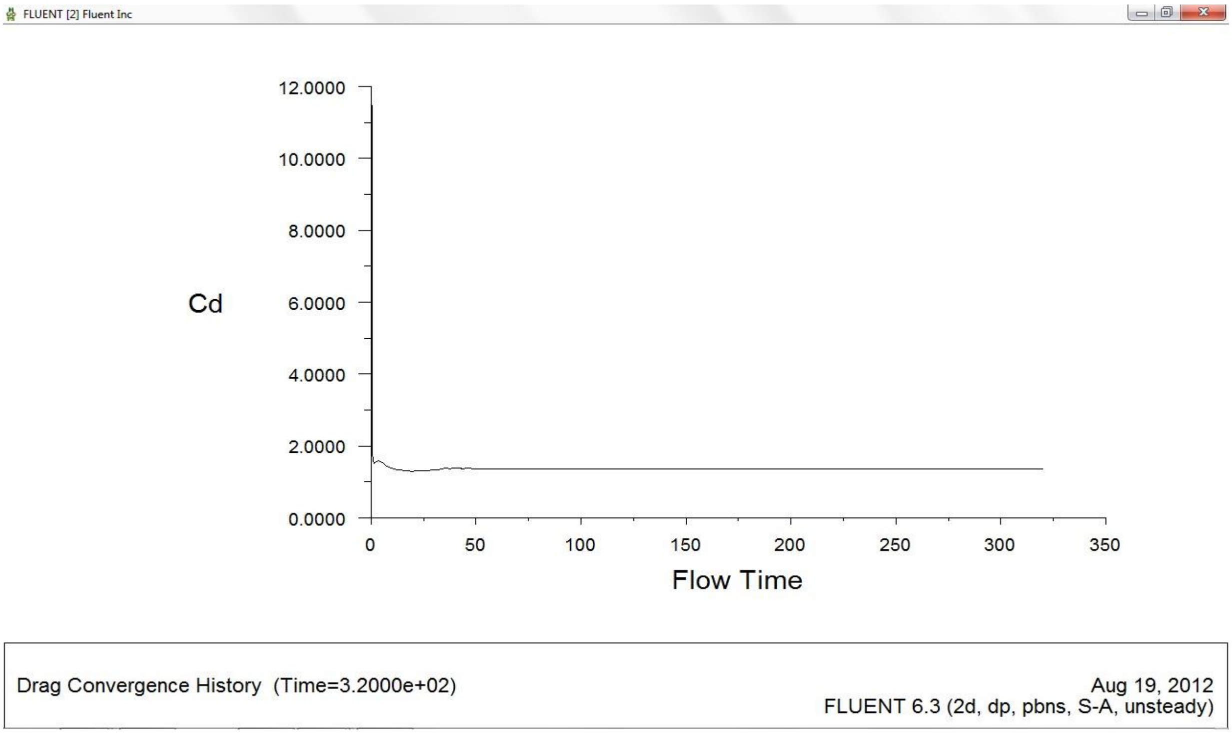


Figure 2: Drag Convergence History

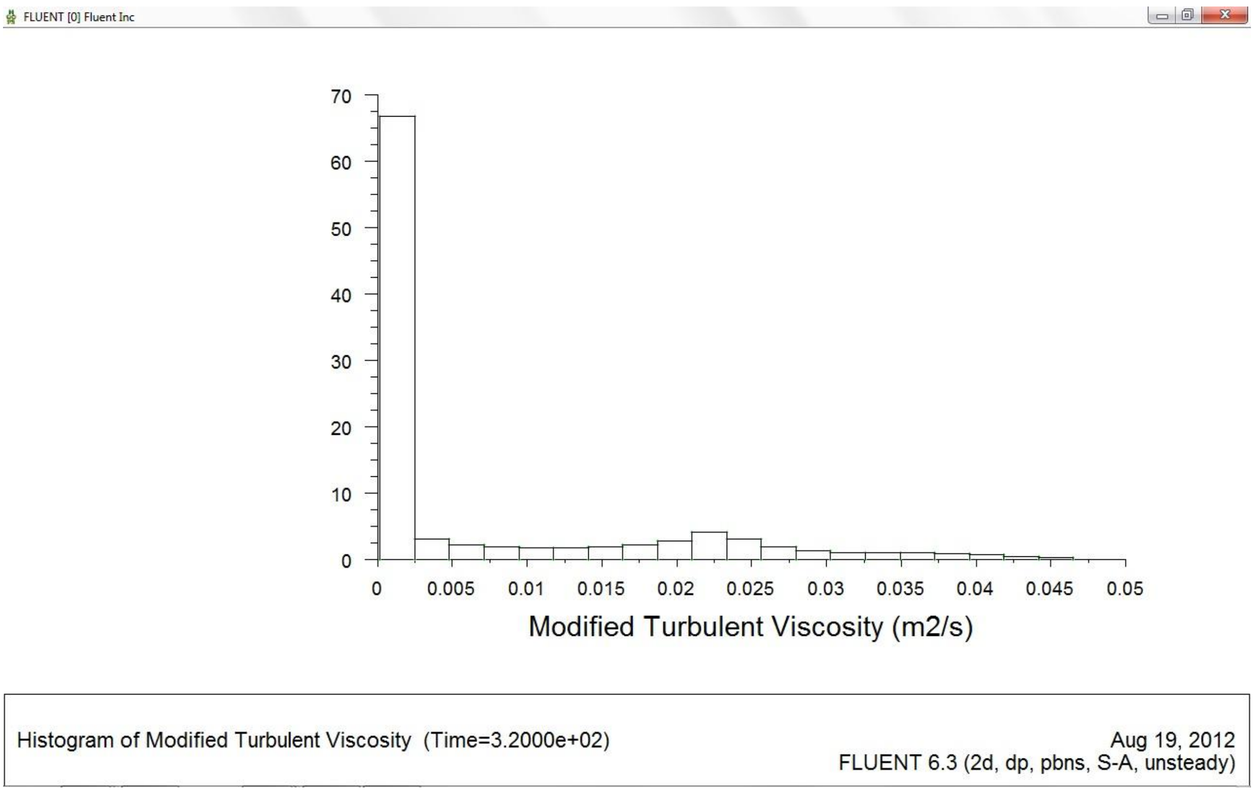


Figure 3: Modified Turbulent Viscosity

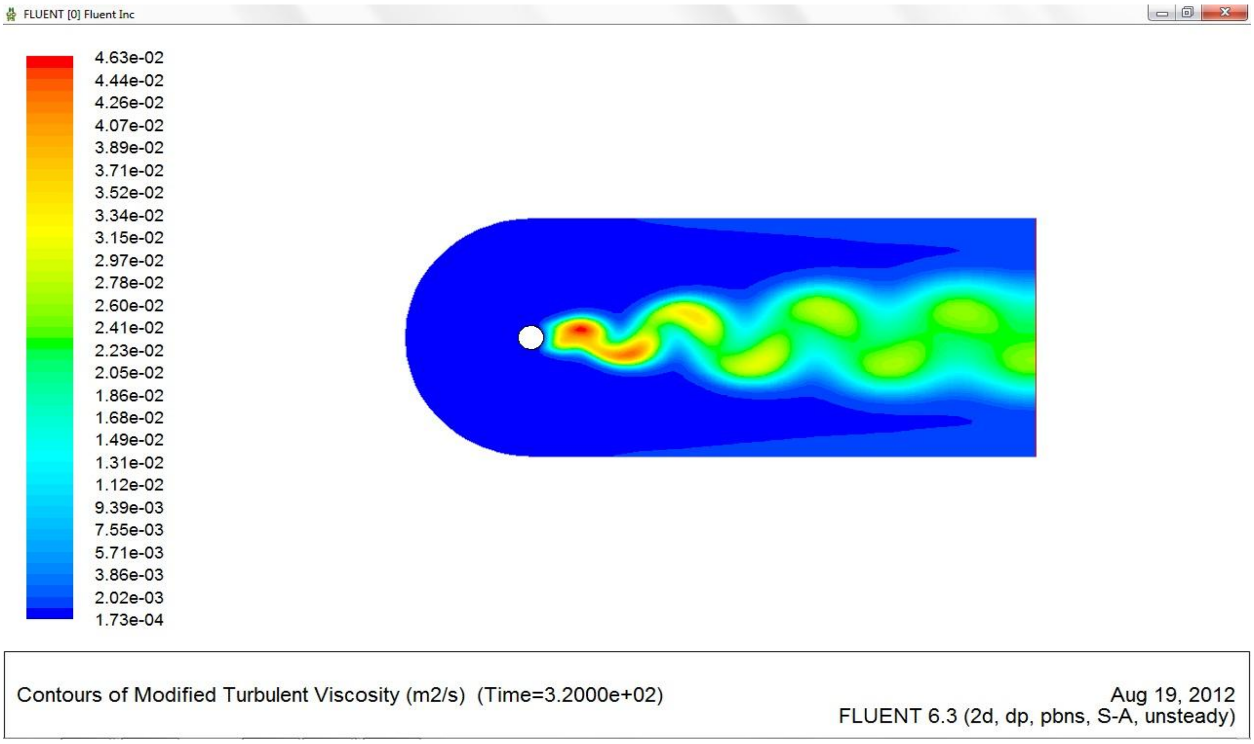


Figure 4: Contour of Modified Turbulent Viscosity

7. Conclusion

After the completion of computational analysis there are following conclusion drawn:

1: drag and lift convergence are being constant, after minimum 60 to70 iteration.

2: turbulence is dense nearby the object, and continuously decreases.

3: maximum pressure held in the front of object.

8. Reference

1. Spalart, P. R. and Allmaras, S. R., 1992, "A One-Equation Turbulence Model for Aerodynamic Flows" AIAA Paper 92-0439

2. Ching Jen Chen, Shenq-Yuh Jaw (1998), Fundamentals of turbulence modelling, Taylor & Francis, http://books.google.co.uk/books?id=HSCUGJ1n8tgC

3. John J. Bertin, Jacques Periaux, Josef Ballmann, Advances in Hypersonic: Modelling hypersonic flows Baldwin B. S. and Barth T. J. A one equation turbulence transport model for high Reynolds number wall bounded owes. AIAA-91-0610.

4. Javaherchi T., Aliseda A, Antheaume S., Seydel J., and Pologye B. Study of the turbulent wake behind a tidal turbine through different numerical models. In 62nd Annual Meeting of the APS Division of Fluid Dynamics, volume 54, 2009.

5. Nait Bouda, N., Schiestel, R., Amielh, M., Rey, C. and Benabid, T., "Experimental Approach and Numerical Prediction of a Turbulent Wall Jet over a Backward Facing Step," International Journal of Heat and Fluid Flow, Vol. 29, 2008, pp. 927-944.

6. Adrian, H., and Law, W. K., "Measurements of Turbulent Mass Transport of a Circular Wall Jet," International Journal of Heat and Mass Transfer, Vol. 45, 2002, pp. 4899-4905.