International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163
Issue 04, Volume 4 (April 2017) (SPECIAL ISSUE) www.ijirae.com

Computational Fluid Dynamics (CFD) - Approach to study Incompressible Boundary Layer flows with Turbulence Models

[1]Darshan, [2]Jagadeesh.Y.J,
3rd semester M.Tech[1], Associate Professor[2],
BMSITM, Bengaluru,

Abstract*- In turbulence models we create mathematical models that describe the flow properties of a flowing fluid. A turbulence model is a computational procedure to close the system of mean flow equations and so that a more or less wide variety of flow problems can be calculated. However, far less precision has been achieved in creating a mathematical model that approximates the physical behavior of turbulent flows. Reynolds stress model is used to get the flow behavior and compared with the theoretical solution and the inferences are drawn.

I. INTRODUCTION

Due to the development in the Computer Aided Design / Computer Aided Engineering technology, now it is possible to analysis the fluid flow problem, in general any engineering problems. The CFD approach has come in this way, when compared to the theoretical and wind tunnel approach. In the present work of flow analysis over cylinder is made for turbulence models and postprocessor velocity contour. Initially the geometry is created and then the fluid domain is discritized by mesh. In the solver and postprocessor its solutions and result are obtained. Basically a theory of CFD-turbulence models and new software tool are applied. This technique may be used for designing of any fluid equipment by using the advantage of CAE. Because of the chaotic-like and apparently random behavior of turbulence, we will need statistical techniques for most of our study of turbulence. CFD is a tool that helps solve a wide range of problems in Fluid Dynamics and Heat transfer. These phenomena are governed by sets of partial differential equations which in most cases have no analytical solution. In addition to the governing equations, we also need the boundary and initial conditions, material properties, and geometrical details in order to completely describe the problem.

II. GOVERNING EQUATIONS AND NUMERICAL PROCEDURE

2.1 CONTINUITY EQUATION

Physical principle:- Law of conservation of mass.

$$\frac{\partial}{\partial t}\iiint_{\upsilon} \rho \, d\upsilon + \iint_{S} \rho \, V dS = 0$$

2.2 DISCRETISATION METHODS

The discretisation methods i.e. the numerical methods for solving Partial Differential Equations include the Finite Difference Methods (FDM), the Finite Volume Method (FVM-also known as control volume method), and the Finite Element Method (FEM). Other methods, such as spectral schemes, boundary element methods and the cellular automata are used in CFD but their use is limited to special classes of problems. Each type of method yields the same solution if the grid is fine enough.

2.3 BOUNDARY CONDITIONS

The present problem has three types of boundaries. They are inlet, outlet and wall boundary. The way these boundary condition are prescribed are:

a) *Dirichlet Condition: Inlet---free stream Velocity boundary condition.*
b) *Outlet – pressure outlet boundary conditions require the specification of static pressure at the outlet boundary-gauge pressure-atmospheric.*
c) *Neymann Condition: Wall – In the present case wall temperature with no slip condition.*

IJIRAE: Impact Factor Value – SJIF: Innospace, Morocco (2016): 3.916 | PIF: 2.469 | Jour Info: 4.085 | ISRAJIF (2016): 3.715 | Indexcopernicus: (ICV 2015): 47.91

2.4. TURBULENCE MODELS

REYNOLDS STRESS EQUATION MODEL

The most complex classical turbulence model is the Reynolds Stress Equation Model (RSM), also called the second order or second moment closure model. Several major drawbacks of the k-ε model emerge when it is attempted to predict flows with complex strain fields or significant body forces. Under such condition the individual Reynolds stresses are poorly represented even if the turbulent kinetic energy is computed to reasonable accuracy. The exact Reynolds stress transport equation on the other hand can account for the directional effects of the Reynolds stress field.

The exact equation for the transport of R_{ij} takes the following form:

$$\frac{DR_{ij}}{Dt} = P_{ij} + D_{ij} - \varepsilon_{ij} + \prod_{ij} + \Omega_{ij}$$

Advantages isVery accurate calculation of mean flow properties and all Reynolds stresses for many simple and more complex flows including wall jets, asymmetric channel and non-circular duct flows and curved flows.Disadvantages isVery large computing costs (seven extra PDEs).

III. ANALYSIS-RESULT

A given set up whose theoretical solution was available was modeled in GAMBIT and then subjected to analysis in FLUENT. A comparative study between Reynolds Stress Model models and theoretical was made, the results of which have been tabulated below. Five hundred iterations were carried out for each of the analysis in Fluent.

3.1. THE EXPERIMENTAL SET UP CONDITION IS AS FOLLOWS:

Atmospheric air at T = 300 K and a free stream velocity of u = 30 m/s flows across a circular cylinder of diameter 3 cm. the surface of the cylinder is maintained at 400 K.

Properties of Air: Density = 1.225 kg/m^3, Specific Heat Capacity, Cp = 1006.43 J/Kg-K

Thermal Conductivity (K)= 0.0242 W/m-K, Viscosity(ν) = 1.789e-5 kg/m-s

3.2. VELOCITY CONTOURS

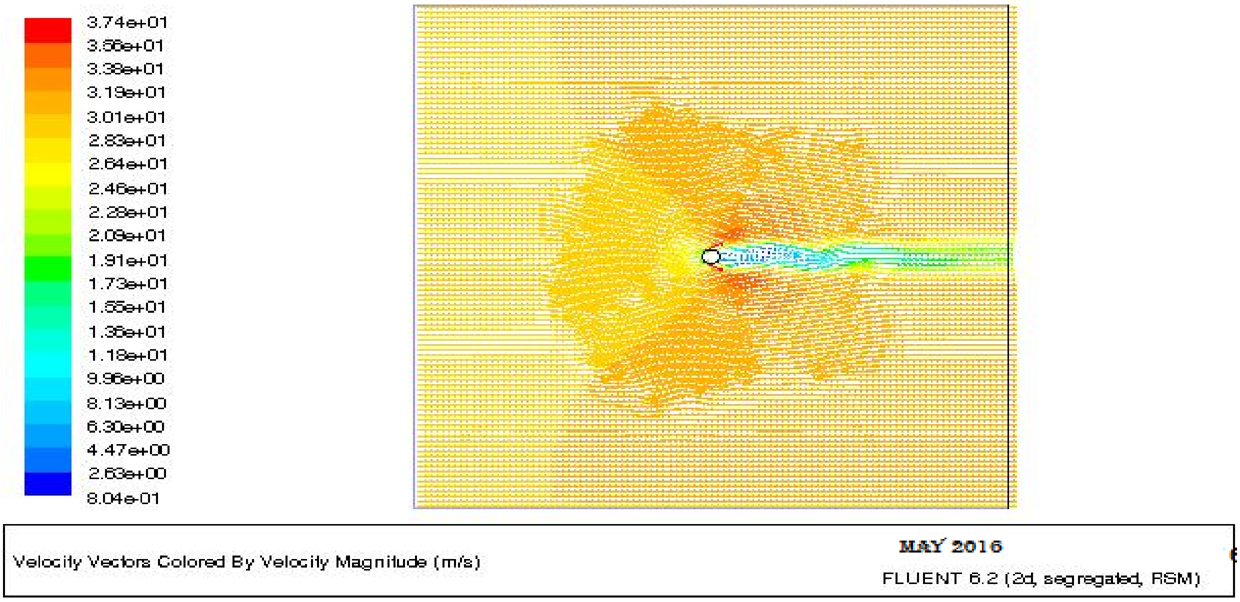

Fig:1. Reynolds Stress Model

3.3. TABULATION OF RESULTS:

	AVERAGE HEAT TRANSFER COEFFICIENT (W/M^2-K)	ERROR %	HEAT TRANSFER RATE (W/M)	ERROR %
THEORETICAL	119.73	-	1128.46	-
FIVE EQUATION MODEL: REYNOLDS STRESS	78.58	34.37	814.72	27.80

INTERPRETATION OF RESULTS:

From the above study of the various models using the FLUENT software it is seen that for the given setup of air flowing across a cylinder the Reynolds stress model agree with the theoretical value.

IV. CONCLUSION

Turbulence is a phenomenon of great complexity and has puzzled theoreticians for over a hundred years. Its appearance causes radical changes to the flow which can range from the favorable (efficient mixing) to the detrimental (high energy losses) depending on one's point of view. The fluctuations associated with associated with turbulence give rise to the extra Reynolds stresses on the mean flow. What makes turbulence so difficult to tackle mathematically is the wide range of length and time scales of motion even in flows with very simple boundary conditions.Although the resulting mathematical expressions of turbulence models may be quite complicated it should never be forgotten that they all contain adjustable constants that need to be determined as best–fit values from experimental data that contain experimental uncertainties.

NOTATIONS:

P-pressure
k- turbulent kinetic energy
K-mean kinetic energy
μ - viscosity
η-eddy viscosity
τ - viscous stress
U,V,W-free stream mean velocity
u',v',w'- instantaneous velocity
ρ – density
e – rate of deformation
E – mean rate of deformation
R- Kinetic Reynold stress
Ω- Rotaion term
$\prod$ -pressure – strain correlation term
D- diffusion
P- rate of production of R

V. REFERENCES

[1]. Schlichting H. (1979), Boundary Layer Theory, Seventh Edition, McGraw Hill
[2]. Cho J.T., Chung M.K. (1992) Intermittency Transport Modeling based on Interactions between intermittency and mean velocity gradients, Journal of Fluid Mechanics, vol. 237, pp. 301-322
[3]. Suzen Y.B., Huang P.G. (2005) Numerical simulation of unsteady wake/blade interactions in low-pressure turbine flows using an intermittency transport equation, Journal of Turbomachinery, ASME vol.127, pp. 431- 444
[4]. Majumdar S.,Rajani B.N.,Kulkarni D.,Mohan S.,(2003),RANS computation of low speed turbulent flow in complex configuration, Proceedings, Symposium on State of the Art and Future Trends of CFD at NAL, NAL SP 03 01
[5]. Patankar S. V. (1980), Numerical Heat Transfer and Fluid flow, McGraw-Hill, New York, N.Y.
[6]. Launder B.E., Spalding D.B.(1974), The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering, vol 3, p. 269
[7]. Introduction to CFD by H.K. Versteeg and W. Malala Shekera, LongMan Sc & Tech.
[8]. http://www.tfd.chalmers.se/˜lada, Department of Thermo and Fluid Dynamics. Chalmers University of Technology, G¨oteborg, Sweden, November 2003 Fluent 6.2 Help Manual
[9]. A κ-ε-r equation turbulence modelby Ji Ryong and Myung kyoon Chung,J.fluid Mechanics 1992.vol.237, pp 301 – 322 www.cfd-online.com

