A Comprehensive Study on Numerical and Computational Aspects of Turbulence Modelling

Yash S. Gujarathi
Department of Mechanical Engineering
K. K. Wagh Institute of Engineering Education and Research
Nashik, India.
yashsg23@gmail.com

Abstract— Computational fluid dynamics (CFD) is the analysis of the system involving fluid flow, heat transfer and transport phenomena such as chemical reaction, combustion by means of computer based simulation.
In the present work, the numerical models, governing equations involved in CFD, Reynolds Averaged Navier-Stokes equation along with the k-epsilon model for modelling of turbulence has been studied numerically and computationally to analyze a fluid flow in Y-duct using ANSYS FLUENT 16.2 as a solver. The flow was analyzed and flow characteristics for output were identified by the CFD solver.

Keywords-*cfd; tubulence; k-epsilon; ansys; fluent*

*****

I. INTRODUCTION

Computational fluid dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reaction by means of computer based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas. Some examples are:

- Aerodynamics of aircraft and vehicles: lift and drag
- Hydrodynamics of ship
- Power plant: combustion in IC engines and gas turbines
- Turbomachinery: flows inside rotating passage, diffusers etc.
- Electrical and electronics engineering: cooling of equipment including microcircuits
- Chemical process engineering: mixing and separation, polymer moldings
- External and internal environment of buildings: wind loading and heating/ ventilation
- Marine engineering: loads on off shore structures
- Hydrology and oceanography: flows in rivers, estuaries, oceans
- Metrology: weather prediction
- Biomedical engineering: blood flows through arteries
- Automobile engineering
- Sports: analysis of sprinter etc.

The ultimate aim of development in the CFD field is to provide a capability comparable with other CAE (computer-aided engineering) tools such as stress analysis codes. The main reason why CFD has lagged behind is the tremendous complexity of the underlying behavior, which precludes a description of fluid flows that is at the same time economical and sufficiently complete. The availability of affordable high performance computing hardware and the introduction to user friendly interfaces have led to a recent sprung of interest and CFD has entered into wider industrial community.

II. NOMENCLATURE

σ_g, σ_k	Closure coefficient
$C_{\epsilon 1}, C_{\epsilon 2}, C_\mu$	Closure coefficient
i,j,k	Unit vectors in X,Y,Z direction
k	Kinetic energy of turbulent fluctuation per unit mass
p	Instantaneous static pressure
u	Instantaneous velocity
u_i, u_j, u_k	Instantaneous velocity in tensor notations
$\acute{u}_i, \acute{u}_j$	Fluctuating velocity in tensor
$\overline{u}_i, \overline{u}_j$	Mean velocity
$\overline{\acute{u}_i \acute{u}_j}$	Temporal average fluctuating velocity
ε	Dissipation per unit mass
ζ	Closure coefficient
μ	Molecular viscosity
μ_T	Eddy viscosity
RANS	Reynolds averaged Navier stoke
ω	Instantaneous angular velocity
ρ	Density

III. HISTORY

From the 1960s onwards the aerospace industry has integrated CFD techniques into the design, R&D and manufacture of aircraft and jet engines. More recently the

methods have been applied to the design of internal combustion engines, combustion chambers of gas turbines and furnaces. Furthermore, motor vehicle manufacturers now routinely predict drag forces, under-bonnet air flows and the in-car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

A. *Turbulence*

Turbulence is explained using Reynolds experiment consisting of water flowing through pipe having colored dye injected in it. At low velocity colored dye forms layers due to regular orderly motion of fluid elements knowns as laminar flow. As the velocity increases dye tend to diffuse and mix with the flow. This random disordered motion of fluid elements is termed as turbulent flow. At low velocity the perturbation due to change in inlet velocity or wall effect do not get amplified, they just die down but at high velocity imperfections gets amplified due to dominance of inertial effect hence perturbations grow. In general viscous effects dampen the disturbances and inertial effects tend to amplify the disturbances.

We characterize this transition from laminar to turbulent flow with the aid of Reynolds number Re which qualitatively represents the ratio of inertial forces to viscous forces.

B. *Imoprtant features of tubulent flow*

- Randomness of transport variable with respect to time/space
- Exchange of momentum involving strong mixing
- Enhanced diffusivity
- Wide range of length scales and time scales

C. *Engergy cascading*

Eddies can be described as lumps of rotating fluid masses. Largest eddy will be of system length scale while smallest eddy will be of molecular length scale. As the larger eddy extracts K.E from mean flow due to disturbances present in the mean flow this energy is evolved to smaller eddy and is dissipated by viscous dissipation. Hence smaller eddy is just good enough to dissipate the energy by viscous dissipation having Re of the order of i.e. inertial forces just balance the viscous forces. This is how energy is cascade in turbulent flows.

IV. RANS

The Reynolds Averaged Navier-Stokes equations (also known as RANS equations) are equations used to predict the fluid flow using a time averaged formulation. The primary concept applied is Reynolds decomposition which involves decomposing an instantaneous quantity into its time averaged of fluctuating quantity. The time averaged nature of its equations makes it an attractive choice while simulating turbulent flows. Considering certain approximations based on the knowledge of properties of turbulent flows, these equations can be used to give time averaged solutions to the Navier–Stokes equations.

$$\rho\left[\frac{\partial \bar{u}_i}{\partial t}+\frac{\partial}{\partial x_j}\left(\bar{u}_i\bar{u}_j\right)\right]=-\frac{\partial \bar{p}}{\partial x_j}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial \bar{u}_i}{\partial x_j}\right)-\frac{\partial}{\partial x_j}\left(\rho\overline{\hat{u}_i\hat{u}_j}\right)$$

This equation is known as RANS (Reynolds Average Navier Stoke) equation in terms of mean flow. The process of time averaging has introduced a new term A. This term A is known as Reynolds Stress or turbulent stress as it has same unit as of stress. This term A is second order stress tensor given by:

$$-\frac{\partial}{\partial x_j}\left(\rho\overline{\hat{u}_i\hat{u}_j}\right)=\begin{bmatrix}\overline{u^2}_1 & \overline{u_1^1u_1^2} & \overline{u_2^1u_3^1}\\ \overline{u_1^1u_2^1} & \overline{u_2^2} & \overline{u_2^1u_3^1}\\ \overline{u_1^1u_3^1} & \overline{u_2^1u_3^1} & \overline{u_3^2}\end{bmatrix}$$

A. *Closure problem in turbulence modelling*

In RANS equation, Reynolds stress terms give six additional unknowns, but there are no explicit governing equations for additional unknowns. There are 3 velocity component, 1 pressure & 6 Reynolds stress terms i.e. total 10 unknowns, whereas number of equations are 4 (1 continuity + 3 components of momentum equation). As the numbers of unknowns are more than number of equations, the problem is in determinant .One needs to close the problem to obtain the solution. This is known as closure problem in turbulence modelling. The turbulence modelling tries to represent the Reynolds stress in terms of time- avg. velocity component. The common turbulence models are classified on the basis of the number of transport equation that needs to be solved along with RANS equation. For different types of turbulent models several approaches have evolved to model Reynolds stress tensor:

1. Eddy viscosity model
2. Reynolds stress transport model

Depending upon the extra number of governing transport equations modelled various methods of modelling Reynolds stresses have evolved. The standard k-epsilon model is most widely used in industrial applications and hence studied and analyzed in the present work.

Mixing length model is governed by zero equations, Spalart Allamars model is governed by one equation, and Standard k-epsilon model and k-w is governed by two equations and Reynolds stress model is governed by seven equations.

TABLE I. TURBULENCE MODEL DEPENDING ON TRANSPORT EQUATION

No. of extra equations	Name of model
Zero	Mixing Length
One	Spalart-Allamars model
Two	Standard k-epsilon model RNG k-epsilon model Realizable k-epsilon model k-w model
Seven	Reynolds stress model

V. K-EPSILON MODEL

The k-epsilon model is one of the most commonly used turbulence models. It is a two equation model that employs two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy. K-epsilon (k-ε) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. This model gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the K-epsilon model was to improve the

mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows.

A. Standard k-epsilon model

The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy (k). The second transported variable is the turbulent dissipation (ε) which determines the rate of dissipation of the turbulent kinetic energy. This model, however, does not perform well in cases of large adverse pressure gradients.

Dissipation rate is given by:

$$\varepsilon = \vartheta \frac{\partial u_i}{\partial x_j}\frac{\partial \bar{u}_i}{\partial x_j}$$

The model equation for turbulent K.E is:

$$\frac{Dk}{Dt} = \frac{\partial k}{\partial t} + \bar{u}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\frac{\vartheta_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + P - \varepsilon$$

The model equation for turbulent dissipation is:

$$\frac{D\varepsilon}{Dt} = \frac{\partial \varepsilon}{\partial t} + \bar{u}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\frac{\vartheta_t}{\sigma_g}\frac{\partial \varepsilon}{\partial x_j}\right] + c_{\varepsilon 1}\frac{p_\varepsilon}{k} - c_{\varepsilon 2}\frac{\varepsilon^2}{k}$$

Fitting model constants are used for modeling approximate equations. The standard values for all model constants as fitted with benchmark experiments (Launder) are given by Cmu=0.09, Cepsilon1=1.44, Cepsilon2=1.92

This is how k-epsilon model is closed.

B. Realizable k-epsilon model

The Standard k−ε is a well-established model capable of resolving through the boundary layer. The second model is Realizable k−ε, an improvement over the standard k−ε model. It is a relatively recent development and differs from the standard k−ε model in two ways. The realizable k−ε model contains a new formulation for the turbulent viscosity and a new transport equation for the dissipation rate, ε, which is derived from an exact equation for the transport of the mean-square vorticity fluctuation. The term "realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard k-ε model nor the RNG k-ε model is realizable. It introduces a Variable Cμ instead of constant. An immediate benefit of the realizable k-ε model is that it provides improved predictions for the spreading rate of both planar and round jets. It also exhibits superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation. In virtually every measure of comparison, Realizable k-ε demonstrates a superior ability to capture the mean flow of the complex structures.

VI. PRE PROCESSING

A. Geometry Creation

The geometry of y-duct isn't much complicated thus ansys design modeler was used for creating the geometry instead of creating the geometry in 3D modelling software and importing it to ansys. Ansys design modeler is inbuilt ansys software for geometry and surface creation. Dimension of y-duct were obtained from hvac application where two fluid streams mix and accordingly geometry was created. Fluid enclosure was created for the solid y-duct where the fluid volume is analyzed to reduce the computational time and cost.

B. Mesh Generation

Generating high quality mesh gives better solver results hence emphasis has to be given on creating a high quality mesh. For that element sizing options were selected and using element size of 0.1 mesh was generated.an additional 5 layers of inflation has been added to improve the quality of results at the walls. After grid generation the quality of mesh was checked using skewness matrix in ansys and the vales of skewness were 0 to 0.4 thus proving the mesh of good quality.

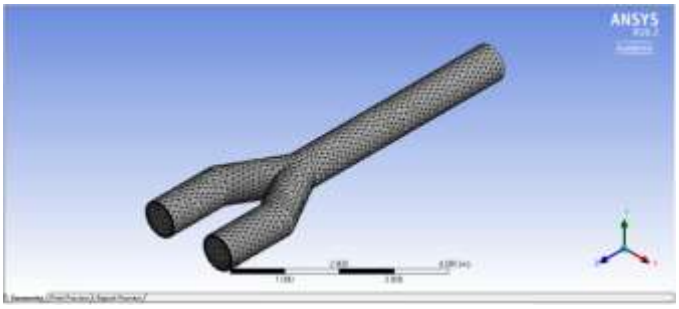

Figure 1. Mesh generated

C. Boundary Conditions

The name selections were done for various planes i.e. for inlet "inlet1" and "inlet2". Similarly for output and for walls name selection was done. Boundary conditions for inlet1 were set to velocity of fluid of 1.5 m/s, temperature of 323.15 K and for inlet2 velocity of 3.5 m/s and temperature of 298.15K .velocity and temperature of fluid at the outlet was observed.

VII. SOLVER

For analysis of fluid flow through y-duct following assumptions were made and according the solver was selected. ANSYS FLUEN 16.2 was used as a solver. Results were obtained for 1000 number of iterations and verified that solution is converging by monitoring the area weighted average temperature and for 1000 iterations the graph of temperature was steady

- Properties of air having density of 1.137 kg/m3, specific heat 1.007 KJ/kg.K, thermal conductivity of air 0.02689 W/m.K and viscosity of air 1.9838* 10^-5 Pa-s
- Reynolds number was calculated and since it was greater than 2000 the flow is turbulent.
- Flow is assumed to be steady
- Pressure based steady state solver has been selected.
- Standard k-epsilon model for turbulence has been selected.

- Turbulent intensity of 1% and hydraulic diameter of 750mm

VIII. Solver Results

The solver results obtained were:

- Velocity of air at outlet: 4.98832 m/s
- Temperature of air at outlet:306.15211 K

A. *Post Processor*

The velocity, temperature and turbulent kinetic energy are shown below

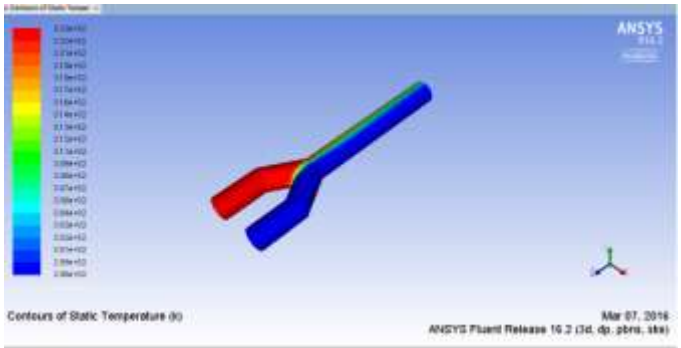

Figure 2. Static temperature contour

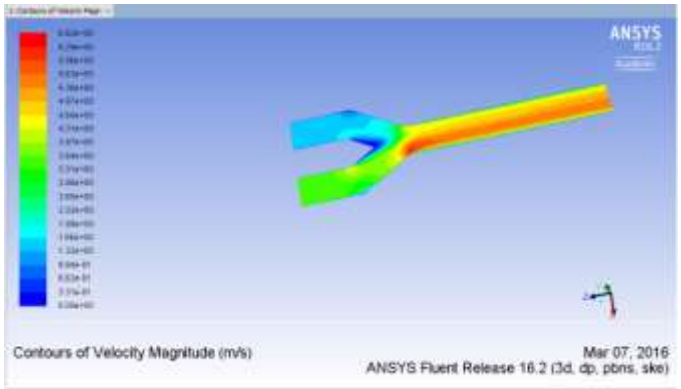

Figure 3. Velocity contour

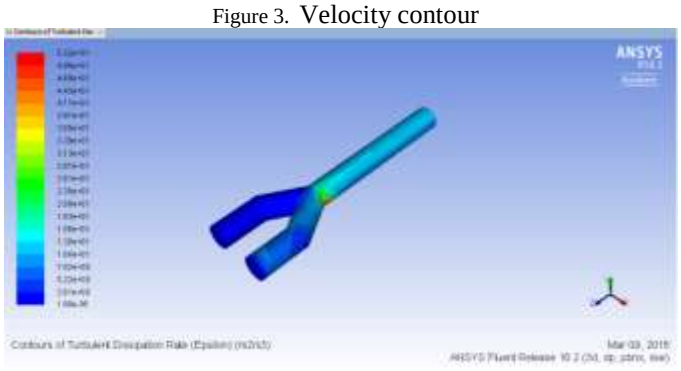

Figure 4. Turbulent dissipation contours

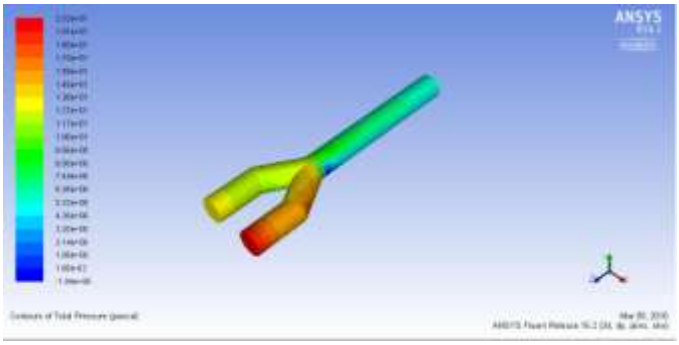

Figure 5. Total pressure contour

IX. Conclusion

Wide range of length scales and time scales of motion makes the prediction of the effect of turbulence more difficult. RANS turbulence model works well in expressing the main features of any turbulent flows by means of one length scales and time scales. The standard k-epsilon is widely used in industrial internal flows computation although it gives poor predictions for swirling and rotating flows, fully developed flows in non-circular ducts, whereas k-w model has become established as leading model for aerospace applications. Hence one model doesn't perform well for all the problems. Fluids flows problems for y-duct can be successfully solved using k-epsilon model with ansys fluent 16.2 as a solver.

References

[1] Henk Kaarle Versteeg, Weeratunge Malalasekera (2007), "An Introduction to Computational Fluid Dynamics: The Finite Volume Method", Pearson Education Limited, ISBN 9780131274983.

[2] H. Versteeg, W. Malalasekera, "An Introduction to Computational Fluid Dynamics: The Finite Volume Method (2nd Edition)",Pearson Education Limited, 2007, ISBN 0131274988

[3] Launder, B.E., and Spalding, D.B,"The numerical computation of turbulent flows",Computer Methods in Applied Mechanics and Engineering 3 (2): 269–289, (March 1974), doi:10.1016/0045-7825(74)90029-2

[4] T. H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu, "A New k-ε Eddy Viscosity Model for High Reynolds Number Turbulent Flows—Model Development and Validation", Computers Fluids. 24(3):227-238, 1995

[5] Anand, R.B., Sandeep, "Reflect of angle of turn on flow characteristics of Y-shaped diffusing duct using CFD", Frontiers in Automobile and Mechanical Engineering (FAME), 2010, 25-27 Nov. 2010.