Journal of Applied Sciences

ISSN 1812-5654

ANSInet
an open access publisher
http://ansinet.com

Journal of Applied Sciences 7 (1): 121-126, 2007
ISSN 1812-5654

A Comparative Assessment of Turbulence Models for Axisymmetric Confined Jet With Back Pressure

[1]Hooman Hajikandi and [2]Abbas Mansoori
[1]Department of Irrigation Engineering, Science and Research Campus,
Islamic Azad University, Tehran, Iran
[2]Department of Hydraulic Structures, Faculty of Engineering, South Tehran Branch,
Islamic Azad University, Tehran, Iran

Abstract: The families of K-ε and K-ω turbulence models are performed for axisymmetric confined jets with back pressure and the results are compared with the experimental data. The physical model which is used as a source to evaluate the numerical results consists of a sudden pipe expansion, with an Expansion Ratio (ER) of 3.675. the results show that the realizable K-ε model and RNG K-ε model give a better prediction of the confined jet spreading rate, while the standard K-ε and the family of K-ω models end in overestimate approximations. However, some models have local superiority to the others, in special zones of the flow or give a better prediction of a class of flow parameters or flow characteristics. So, the authors ignore setting forth a special turbulence model for general implementation in numerical calculations for confined jets, but point out the efficiencies and deficiencies of each model.

Key words: Turbulence models, jets, confined flow, separation

INTRODUCTION

Turbulence flows are characterized by fluctuating components, which have three dimensional behaviors and a large spectrum of length scales. The special stochastic characteristics of turbulent elements, together with flow unsteadiness and three dimensional nonlinear oscillations, add to the complexity of the problem. The governing equations of turbulence are the Navier-Stokes equations. Due to numerous unknowns and nonlinearity of these equations, up to now, no analytical solution except for a few simple homogeneous, isotropic situations, has been proposed. So, the numerical approaches are the only available tools to be implemented in computational codes.

Engineering CFD calculations are largely based on the solution of the RANS equations in conjunction with a turbulence model. During last decades, many different turbulence models have been proposed by different researchers.

The models proposed so far are often based on theoretical principles, dimensional analysis, asymptotic analysis of the Navier Stokes equations and experimental observations.

Until now despite the efforts and available assessments and general consensus there is no single theory and a dominantly superior turbulent model which can be used reliably and universally. Often, turbulent models are developed and tested for limited classes of flow. The first turbulence models of which, the prandtls mixing length hypothesis is the most well known example, related the turbulent transport terms uniquely to local mean quantities. These models could only be applied to a self similar special flow groups, which the partial differential equations expressing mass and momentum conservation can be reduced to ordinary ones. Even for the few simple flows which the zero equation models could be applied, the early models lacked the universality in that they required different empirical constants for different flows. The mixing length model is not suitable when turbulent convective transport is an important part of the physics, examples are rapidly developing flows, heat transfer across planes with zero velocity gradients and flows with recirculation. Also due to the great difficulties in specifying the mixing length (L), the model is of limited use in complex flows.

In order to overcome the above mentioned limitations of the mixing length hypothesis, turbulence models were developed which account to the transport of turbulence equations by solving differential transport equations for them. Among these models the one equation model of Kolmogorov-Prandtl, which was introduced by Kolmogrov (1942) and Prandtl (1925) independently, is well known.

Corresponding Author: Hooman Hajikandi, Faculty of Civil Engineering, Islamic Azad University, Central Tehran Branch, Tehran, Iran Tel: 009821-88370651

The more advanced models developed in 1940 s, give up direct algebraic link between the turbulent transport terms and the mean flow quantities and employ differential transport equations for turbulence quantities such as kinetic energy of the turbulent motion. In 1960 s computers became sufficiently powerful and shortly after, numerical techniques developed to allow partial differential equations to be solved for many flow situations.

The most popular turbulent models which have been developed since 1980 s, are the standard K-ε model proposed by Launder and Spalding (1972), Realizeable K-ε model of Shih *et al.* (1998), RNG K-ε model introduced by Choudhury (1993), standard K-ω model of Wilcox (1988) and its shear transport version of Menter (1994). All of the above models have some limitations. For example, the widely used K-ε turbulence model fails to predict flow near a solid boundary. It also fails to predict the experimentally observed differences between the spreading rates of a plane jet and a round jet. In experiments the spreading rate of round jet is 15% lower than a plane jet, but in the simulations with the standard K-ε model, the round jet spreading rate is 15% higher (Wilcox, 1988). Wilcox proposed that the dissipation rate equation of the K-ε model be replaced by an equation for a specific dissipation rate defined as $\omega = K/\varepsilon$. This K-ω model predicts the behavior of attached boundary layers in adverse pressure gradients more accurately than K-ε model, but performs weakly in free shear flows.

In 1982, Sinder (1982) compared the application of common turbulent models and proposed some corrections for the coefficients of the K-ε model. Kooronaki *et al.* (2001) investigated the effect of the kind of turbulent model on the results obtained for velocity and turbulent distribution. Cole *et al.* (2003) simulated axisymmetric submerged jets numerically using FLUENT CFD code and pointed out that the results obtained for velocity distribution profiles match well with experimental data.

One of the most ambiguous categories of flow is the confined jet with back pressure. The flow separation phenomena with recirculating eddies, make the problem too intricate. The present work is concerned with a comprehensive assessment of five common two equation turbulent models, namely: standard K-ε, RNG K-ε, Realizable K-ε, standard K-ω and SST K-ω implemented in general purpose FLUENT V6 CFD code. Each model is run with its standard set of constants. The numerical method is based on a cell-centered, unstructured finite volume discritization and a SIMPLE type segregated solution procedure. All simulations are carried out to use as second order upwind scheme for the convective terms.

Figure 1 shows the model geometry, which is a two dimensional axisymmetric sudden pipe expansion. Inlet velocity and outlet pressure are chosen as boundary conditions. The diameter of the inlet jet and the expanded pipe diameter are 7.62 and 28 cm, respectively, which produce an expansion ratio of D2/Dj = 3.675. A circular orifice with a diameter of 19.5 cm is provided at the outlet inorder to ensure flow submergence. A uniform rectangular grid, consisting of 13250 quadrilateral element cells, is implemented in the numerical procedures.

EXPERIMENTAL RIG AND INSTRUMENTATION

The expansion used here was located 2.5 m from the inlet of the test section and was preceded by a short, smooth contraction converting upstream pipe diameter of 300 mm to downstream diameter of 76.2 mm. The pipe diameter at the inlet to the expansion was $D_J = 76.2$ mm and its length was 1 m, which ensured formation of fully developed flow at the jet inlet. The main body of the sudden expansion, the key dimensions of which are shown in Fig. 1 and 2, was made of Perspex. The smooth contraction was fabricated form stainless steel. Measurements of pressure and velocity distribution were possible form 11 axial locations corresponding to X values of -6, 6, 12, 18, 24, 30, 42, 54, 66, 78, 114 cm, measured from the efflux section. Also, pressure tappings of 2 mm diameter were provided along the bottom of the pipe to

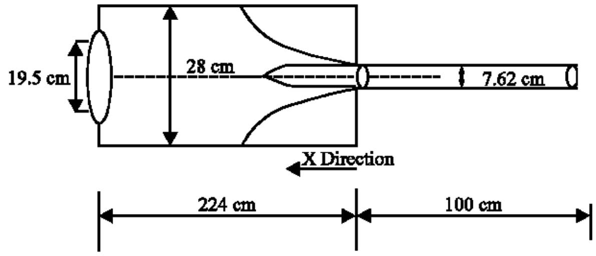

Fig. 1: Sketch of the physical model

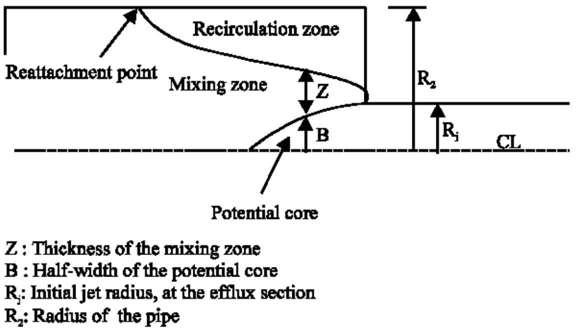

Fig. 2: Defition of flow parameters

allow the wall pressure distribution to be measured. The tapings were connected to differential pressure transducer and the measurements were carried out using a data logger. The pressure transducers used, were able to measure gage pressures between -1 bar and +1 bar with a natural frequency of 1 KHz. Figure 3 shows the photograph of the rig.

Fig. 3: Photograph of the physical model

COMPARISON OF THE RESULTS

Figure 4 compares the results obtained from CFD models for spread of the jet, with experiments. It seems that realizable K-ε model (R K ε) gives the best prediction for the spread of the jet, while with the RNG K-ε model, the spreading of the jet for the zones near reattachment point is 10% lower, however for the zone of flow establishment the results are approximately identical to simulations with the R K ε model. In simulations with the standard K-ε model and K-ω model, the spreading rate of the jet is higher than experimental measurement. This deviation from measurements is more sensible for the zone of flow establishment.

In Fig. 4, both the inner and outer limit of diffusion is illustrated. Comparison of the curves, shows that the family of K-ω models approximate the length of the potential core much less than measurements. Figure 5 shows the results obtained for centerline velocity distribution. As it is obvious, predictions of the RNG model and standard K-ε model for center line velocity distribution match reasonably well with measurement, while the predictions of other models differ considerably from experiments. It is interesting that the simulations with the standard K-ε model provide the closest results to the experiments, for centerline velocity distribution. Distribution of axial velocity at three different sections is shown in Fig. 6. It is seen that the standard K-ε model

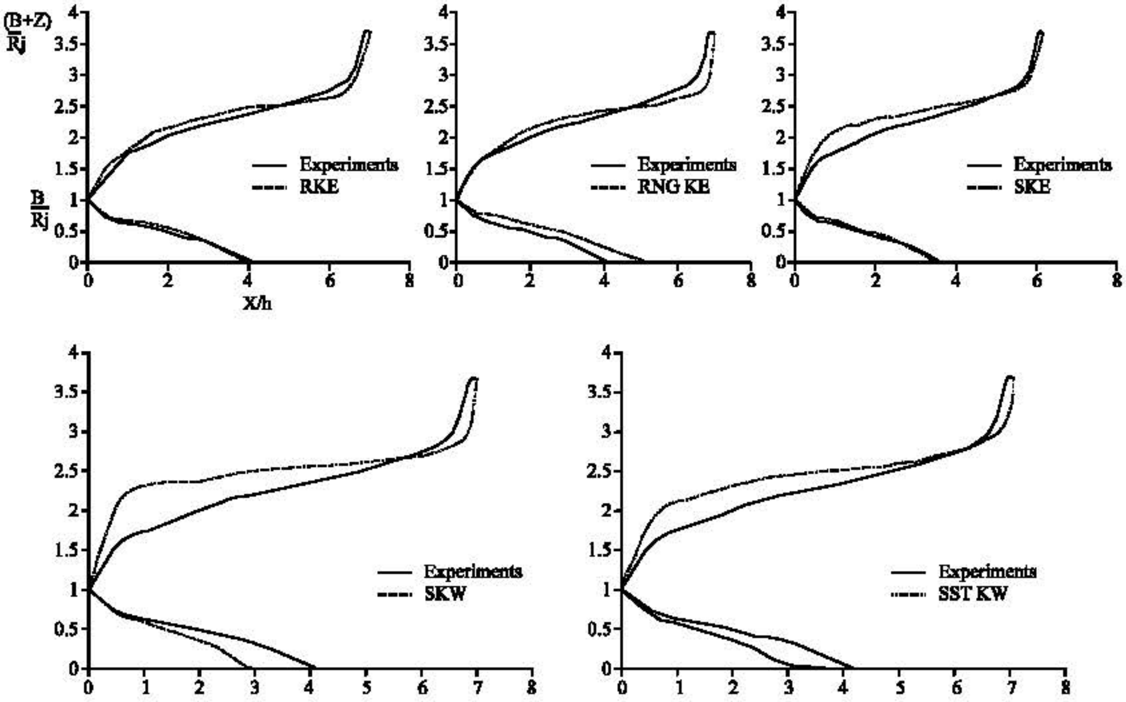

Fig. 4: Comparison of the CFD results for the spread of the jet

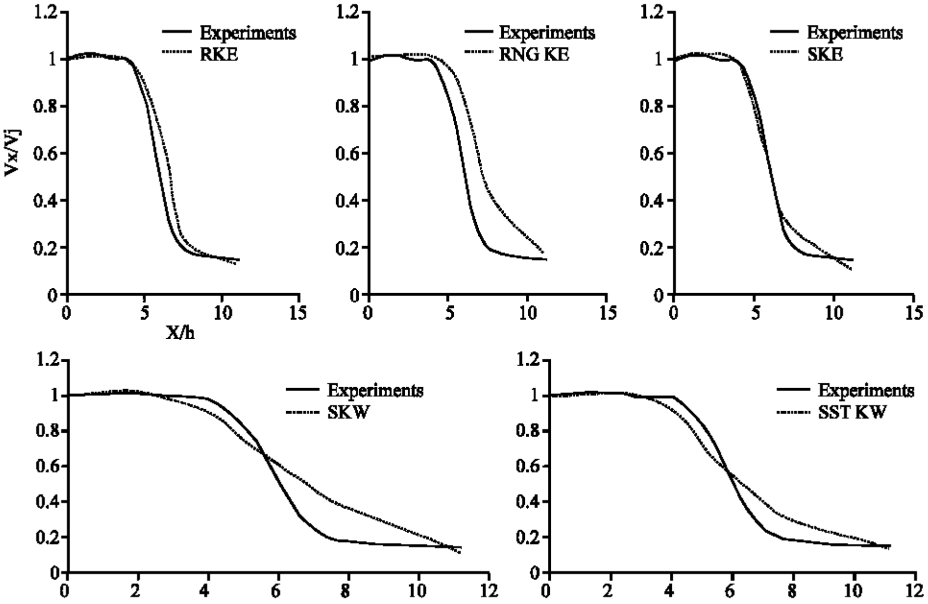

Fig. 5: Distribution of maximum velocity along the pipe (Centerline velocity), RKE: Realizeable K-ε model, RNG KE: Renormalization group K-ε model, SKE: Standard K-ε model, SKW: Standard K-ω model, SST KW: Shear Stress Transport K-ω model

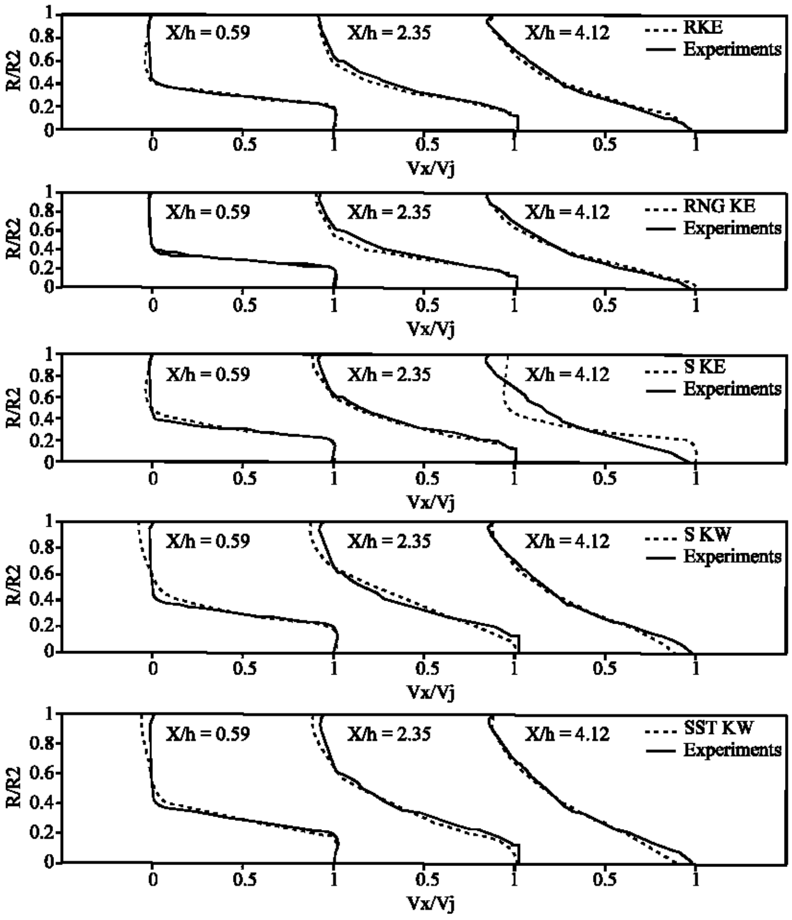

Fig. 6: Axial velocity distribution at three distinct locations

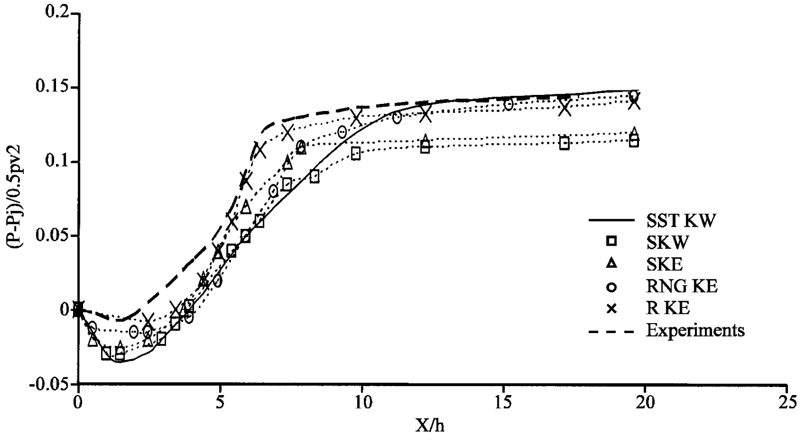

Fig. 7: Distribution of mean static pressure

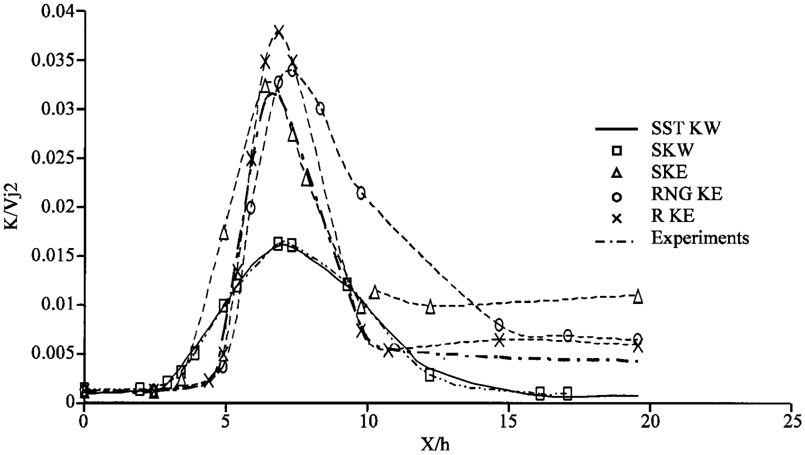

Fig. 8: Comparison of the CFD results for turbulence kinetic energy with experimental results

performs weak in recirculation zone. The mean static pressure distribution is shown in Fig. 7. It is clear that the RK ε model gives the best prediction of static pressure distribution along the jet axis. Finally the comparison of turbulence kinetic energy is presented in Fig. 8. It seems that the RK ε model approximates the trend of the turbulence kinetic energy much better than the other models, while the standard K-ε model and the RNG model predictions of peak kinetic energy match with experimental data, much better than the predictions of other models. On the other hand in simulations with the family of K-ω models, the peak kinetic energy is much lower than experiments.

CONCLUSIONS AND FUTURE WORK

The comparative analysis of the performance of five common 2-equations turbulence models for simulations of axisymmetric confined jets dominated by flow separation has been presented. The objective was to evaluate the overall accuracy of each model to prediction different flow parameters at different flow zones. The results indicate that each model has some limitations and the authors hesitate to introduce a definite model as the best applicable model for simulations of this kind of flow but believes that, overall, the accuracy of the RKε model and RNG K-ε model appears to be superior to other models. Of course, the RNG K-ε model in comparison to the RKε model needs more computational efforts per iteration and for the convergence of the solutions, which is due to the strong nonlinear terms existing in its transformed equations.

The sensitivity of the models to the choice of boundary type and the Reynolds number are the topics which need further investigation and will be the future work of the author.

ACKNOWLEDGMENT

The physical model was constructed in the laboratory of Iran's Water Research Center and special tanks are given to Dr. Nasiri, Head of the Institute for his valuable help.

REFERENCES

Choudhury, D., 1993. Introduction to the Renormalization Group Method and Urbulence modeling, Fluent Inc., Technical Memorandum, Tm-107.

Cole, D.R., G. Ahmadi, M. Glause and W. Kvasnak, 2003. An experimental and numerical study of turbulent flow in an axisymmetric sudden expansion. APS Conference, GM.02

Kolmogrov, A.N., 1942. The Local Structure of Turbulence in Incompressible Viscous Fluid at Very Large Reynolds Numbers. C.R. Acad. Sci., URSS, 32-16.

Kooronaki, E.D., H.H. Liakos, M.A. Founti and N.C. Markatos, 2001. Numerical study of turbulent diesel flow in a pipe with sudden expansion. J. Applied Math. Modeling, 25: 96-104.

Launder, B.E. and D.B. Spalding, 1972. Lectures in Mathematical Models of Turbulence. Academic Press, London, England.

Menter, F.R., 1994. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA J., 32: 32-45.

Prandtl, L., 1925. Bericht Uber Uterssuchungen Zur Ausgebildeten Turbulentz. J. Angew. Math. Mech., 5: 136.

Shih, T.H., W.W. Liou, A. Shabbir and J. Zhu, 1998. A new K-ε eddy viscosity model for high Reynolds numbers turbulent flows model development and validation. Computers J. Fluids., 3: 66-93.

Sinder, M.M.S., 1982. A numerical study of turbulent flows in backward facing step geometries: A comparison of four models of turbulence. Ph.D Thesis, University of California, Davis, California.

Wilcox, D.C., 1988. Reassessment of the scale determining equation for advanced turbulence models. AIAA J., 32: 103-123.