IMPLEMENTATION AND VALIDATION OF DIFFERENT $k$-$\varepsilon$ TURBULENCE MODELS IN ENGINEERING APPLICATIONS

Ashraf Balabel and Wageeh El Askary

Mechanical Power Engineering Department, Faculty of Engineering, Menofia university, Shebin Elkom, Egypt,
E-Mail: ashrabalabel@yahoo.com

(Received March 2009 and accepted June 2009)

In the present paper, a profound investigation of the performance of several $k$-$\varepsilon$ turbulence models variants is carried out. Three $k$-$\varepsilon$ based turbulence models are considered, namely; the standard $k$-$\varepsilon$ model, the $v^2-f$ model and the non-linear $k$-$\varepsilon$ model. The selected turbulence models are validated against simple as well as complex engineering applications to underpin knowledge about the solution accuracy obtained from two-equation turbulence models. Three engineering applications are chosen and numerically simulated based on the control volume formulation, namely; turbulent free jet, turbulent jet impinging on a flat plate, and turbulent separated flow through sudden expansion. In order to validate the numerical results obtained and to investigate the performance of the turbulence models considered, different experimental measurements are used. The present work is primarily motivated by the desire to provide a rational way for deciding how complex a turbulence model is required for a given engineering application and to find out how accuracy changes with model complexity. Due to the superior predictive performance of the modern turbulence models in a wide range of complex engineering applications, it was believed that; 'universal' turbulence model might exist. In general, that is not true. Simple flows can be analyzed using algebraic eddy viscosity models or standard two-equation models. However, in much complicated flows, such as separated flows or impinging problems, a more elaborate level of modeling is required. In such context, the nonlinear and the $v^2-f$ models are appropriate for predicting the inhomogeneous near wall region and the anisotropic Reynolds stresses which is a vital part of flow prediction.

Keywords: Numerical simulation, turbulence modeling, free jet, impinging problem, separated flow.

1. INTRODUCTION

Turbulent flow is of central importance to many engineering applications, e.g. the aerospace industry, process engineering, internal combustion engines and environmental engineering. In general, turbulent flows are computed either by solving the Reynolds-averaged Navier-Stokes equations with appropriate turbulence models for modeling turbulent fluxes or by computing the fluctuating quantities directly. Indeed, it is often concluded that the turbulence modeling and simulations is the pacing item for the rate of progress.
of CFD in fluids engineering. That is reflected by the enormous amount of research on turbulence predictions over the past three decades. Despite great efforts by research engineers and scientists, turbulent flows are not well-understood and remain an unsolved problem in fluid dynamics[1]. Consequently, new concepts and hypothesis are continually being introduced.

The main goal in turbulence research is to obtain the velocity field as a function of space and time for a specific turbulent flow and also to derive scalar fields of some flow property of interest. Numerical predictions of turbulent flows have been limited in accuracy partly by the performance of three key elements, viz. grid generation, algorithm development and turbulence models. It is known that a well constructed grid and a well designed algorithm greatly improve the quality of the solution, and conversely, it is a major contributor to poor results. In addition, numerical simulations difficulties such as the lack of convergence to a desired level are often referred to the poor grid quality or invalid algorithm. However, the turbulence models' accuracy has a big influence on the prediction of the physical phenomena encountered. Consequently, turbulence modeling has a decisive influence on drag, species transport, heat transfer, vorticity distribution and separation. The major difficulty of turbulent flow is that it has a much wide range of length and time scales and high-frequency fluctuations embedded which makes the equations governing turbulent flows to be much more difficult and expensive to solve.

In general, three levels of modeling are commonly adopted in turbulence simulations: the linear eddy-viscosity models (LEVM) which are based on Boussinesq assumption, the Reynolds stress transport model (RSTM) with the six components of Reynolds stress defined by their transport equations and the so-called large-eddy simulation (LES) where 'slowly varying' components are assumed to extend down to the mesh size where sub-grid models, based on the eddy-viscosity concept, are applied. In all approaches, there are obvious limitations and different weaknesses. However, the mixing length concept for eddy viscosity[2] has been remarkably successful for simple boundary layer flows and a notable success for the $k-\varepsilon$ model has been obtained in the definition of the mean flow properties accurately for attached unsteady boundary layers[3]. In the case of separated flows, the capabilities of different turbulence models are less certain. The LES approach is probably realistic away from a solid boundary, but near to a boundary has all the limitations of a simple eddy viscosity models. In addition, LES is a 3D, and hence a very computationally demanding.

The RSTM offers the potential for far more reliable turbulence production. This is because important production terms are calculated directly from resolved variables, while the production terms of two-equation models are modeled. However, the main difficulties in the RSTM are the modeling of pressure-strain terms and the near-wall turbulence. Also, for RSTM, numerical stability is often problematic and a small time step is required for stability. So, it can also be numerically challenging and computationally expensive. All these characteristics of RSTM are regarded as important limitations in the context of the industrial CFD. This has thus motivated efforts to improve LEVM or to construct models which combine the simplicity of eddy viscosity formulation with the superior fundamental strength and predictive properties of second-moment closure. These efforts have given rise to different groups of modified EVM and non-linear eddy viscosity models (NLEVM).

Some improvements of EVM are carried out by introducing ad-hoc corrections, usually to the length-scale equation, and/or the formulation of alternative equations for different length-scale parameters (e.g. dissipation rate $\varepsilon$, vorticity $\omega$, and time scale $\omega^{-1}$). However, none of these addresses the fundamental limitations arising from the unrealistic constitutive relations[4]. Additionally, these approaches are also known to be afflicted by major weakness. The recent novel attempts to improve EVM are focusing on near-wall modeling and nonlinear constitutive relations used in the nonlinear models. The near-wall modeling needs the development of low-Reynolds stress transport models. The basic idea is based on the implementation of the near-wall viscous effects by damping the turbulent viscosity toward a wall with introducing a damping function. Many versions of such models can be found in [5]. The low-Reynolds $k-\varepsilon$ model[6] dramatically overpredicts the heat transfer at the stagnation point. In a more recent research[7], it is concluded that the low Reynolds stress transport models are critical for heat transfer predictions. However, the widely-used standard $k-\varepsilon$ turbulence modeling with its improved low-Reynolds number form near the walls has been recently implemented by our research group in simulating a turbulent flow inside a combined bend-diffuser configuration[8]. The obtained results showed the ability of the turbulence models adopted in predicting the performance of the bend-diffuser system considered.

The most important defects of EVM are the modeling of the Reynolds stresses using the linear Boussinesq stress-strain relations. This gives a wholly unrealistic representation of normal stresses anisotropy, observed in virtually all shear flows, and the substantial errors in complex strain, in which gradients of the normal stresses contribute significantly to the momentum balance. Other predictive deficiencies include incorrect sensitivity to curvature strain and dilation, excessive levels of turbulence in regions of strong normal straining, wrong response to swirl and the suppression of self-induced periodic motions.

To avoid the related problems of EVM, an alternative route thus pursued recently has been the nonlinear eddy-viscosity models (NLEVM)[9]. This
Implementation and Validation of Different k-ε Turbulence Models in Engineering Application

approach is based on the nonlinear extension of the linear stress-strain relation. Consequently, the Reynolds stresses are defined by algebraic formulae depending on nonlinear combination of mean strain and vorticity as well as $k$ and $ε$. In this way, some of the weaknesses of EVM related to turbulence production near stagnation, insensitivity to curvature, separation on curved surface and often poor prediction of transition are avoided and the important, well-defined, effects are represented.

Among several EVM, the standard (STD) $k-ε$ turbulence model\textsuperscript{[19]} is still the most widely used in industrial and engineering applications as it represents a good compromise between accuracy and computational efficiency. It was developed; calibrated and validated to cover a wide range of engineering applications. It is a robust two-equation turbulence model and it yields quite reasonable results in high Reynolds number flows when its restrictions are taken into consideration\textsuperscript{[11]}. Therefore, the two-equation STD $k-ε$ model has been the subject of much research in the last years even though it fails to predict correctly a number of complex flows. In particular, the two-equation STD $k-ε$ model has limitations when the cross-sectional flow area changes or in the impinging problem. Consequently, there is a need to improve the predictions of the STD $k-ε$ model in such applications. To this end, development has recently focused attention on extending the validity of the STD $k-ε$ model in order to provide a wider range of problems with a quality turbulent flow solution at an equivalent cost.

Some improvements of STD $k-ε$ model have been developed through either re-evaluation of the model constants, e.g. the RNG $k-ε$ model\textsuperscript{[12]} or by introducing ad hoc damping function, e.g. the low-Reynolds-Number $k-ε$ model\textsuperscript{[13,14]}. These modifications are adjusted to make the model fit experimental or computational data\textsuperscript{[15]}. Nevertheless, no pretense has been made that any of these models can be applied to all turbulent flows: such as ‘universal’ model may not exist. Each Model has its adv-/disadvantages, limitations and appropriate flow regimes. One of the most important controllers of the turbulence models performance is the estimated values of the model coefficients, which can be only evaluated by advanced experimental measurements over wide range of engineering applications.

An alternative route is used for improving the STD $k-ε$ model near solid boundaries through the evolution of suitable velocity and time scales of turbulence. It was noted that the unacceptable results obtained by STD $k-ε$ model when it is integrated to solid boundaries are due to the assignment of $k$ as velocity scale\textsuperscript{[16,17]}. This and other considerations motivated the $ν^2-f$ model\textsuperscript{[18]}. In this model, the appropriate velocity scale for turbulent transport toward the wall is the velocity fluctuation normal to streamlines $ν$, thereby introducing effects of streamline curvature in a natural way. Moreover, the $ν^2-f$ model formula provides the right scaling in representing the damping of transport close to the wall.

The $ν^2-f$ model originally was developed for attached or mildly separated boundary layers. Recently, it was also applied in examples of massive separation and unsteady vortex shedding. The starting point of the $ν^2-f$ model is the Boussinesq assumption as well as any other linear eddy viscosity model, i.e. the overall effect on the mean flow is completely isotropic. Therefore, more recently, a variety of modifications have been introduced in order to derive non-isotropic relationship for the normal Reynolds stresses. These efforts have given rise to the group of nonlinear eddy-viscosity models in more complex engineering applications.

The NLEVM is an important topic in recent modeling of turbulent flow. This approach has been developed to describe the flow in more physically consistent manner. The NLEVM’s are made to mimic the physics of turbulence by means of mathematical artifacts and calibration, and, in addition, it provides a mechanism for anisotropy of the normal stresses. The basic idea of NLEVM is the extension of the original linear stress-strain relation to a nonlinear form. Consequently, the Reynolds stresses are defined by algebraic formulae depending on nonlinear combination of mean strain and vorticity as well as $k$ and $ε$, with coefficients which may be tuned for particular applications. The NLEVM’s performance was tested in our previous research (El-Askary and Balabel\textsuperscript{[19]}), for asymmetric divergent channel flow, and good results were obtained.

Most NLEVM’s are usually quadratic or cubic according to the degree of the characteristic time scale normally given as the ratio of $k$ and $ε$ ($k/ε$). These differences in the order are of considerable significance. In particular, the cubic fragments play an essential role in capturing the strong effect of curvature on the Reynolds stresses, while the quadratic terms are responsible for the ability. The comparison of many quadratic stress-strain relations with experimental data showed that none achieves much greater width of applicability. However, the model of Craft et al.\textsuperscript{[20]} has proposed a cubic relation between the strain and vorticity tensor and the stress tensor, which does much better than conventional eddy-viscosity models in capturing the streamline curvature and the anisotropic stresses over a wide range of flows.

More recently, linear and non-linear eddy viscosity models have been applied for more complex cases considering the numerical simulation of the flow and heat transfer from a row of round jets impinging onto a concave semi-circular surface\textsuperscript{[21]}, designed to reproduce flow characteristics found in internal turbine blade cooling applications. It is found that, both of linear and non-linear eddy viscosity models broadly reproduce the mean flow field. Although the non-linear model produced slightly better turbulence stress
levels, both linear and non-linear models underpredicted the levels of turbulence energy as the jets approached the curved surface.

In the present paper, the STD k-ε, the ν²-f and the nonlinear k-ε models are investigated against three axisymmetric flow problems; namely: free round jet, the impinging jet problem, and the turbulent separated flow through sudden expansion, to underpin knowledge about the solution quality obtained from different assorted formulations of two-equation turbulence models in different engineering applications.

The sections below give details of the governing equations and the different turbulence modeling strategies and the numerical approaches adopted. Furthermore, the cases studied are explained in details and the present numerical results obtained are compared with different experimental measurements. Finally, conclusion is drawn.

2. COMPUTATIONAL METHODOLOGY

The Governing Equations

The numerical method adopted here is based on solving the Reynolds-averaged Navier Stokes (RANS) equations. In tensor notation, the RANS equation for incompressible flows can be written as follows:

\[
\frac{\partial}{\partial x_i} (\rho u_i) = 0 \tag{2.1}
\]

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{1}{3} \frac{\partial}{\partial x_j} \left[ \rho (\delta_{ij} \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}) - \frac{2}{3} \rho k \delta_{ij} \right] \tag{2.2}
\]

where \( u_i, u_j \) denote ensemble-average quantities and \( u' \) is the fluctuating or turbulence quantities, \( p \) is the pressure and \( \rho \) is the density. The last term in the above equation represents the Reynolds stresses. In order to close the above momentum equations, turbulence models are needed to model the apparent Reynolds stresses. For the so called linear EVM, the linear stress-strain relation is approximated using the Boussinesq assumption:

\[
-\rho u' u' = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \tag{2.3}
\]

where \( \delta_{ij} \) is the Kronecker delta function (\( \delta_{ij} = 1 \) if \( i=j \) and \( \delta_{ij} = 0 \) if \( i \neq j \)), \( k \) is the turbulent kinetic energy and \( \mu_t \) is the turbulent viscosity.

Turbulence Models

The transport equations for the turbulent kinetic energy \( k \) and its dissipation rate \( \varepsilon \) are:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho u_j k) = \frac{\partial}{\partial x_j} \left[ \mu + \mu_t \frac{\varepsilon}{k} \right] \frac{\partial k}{\partial x_j} + \frac{\varepsilon}{3} + \frac{\partial}{\partial x_j} \left( \mu + \mu_t \frac{\varepsilon}{k} \right) \frac{\partial u_j}{\partial x_j} \tag{2.4}
\]

\[
\rho (P_e - \varepsilon) \tag{2.5}
\]

\[
\rho (C_{f1} f_P - C_{f2} f_P^2) \frac{\varepsilon^2}{k} + \rho \varepsilon \tag{2.6}
\]

The homogeneous and inhomogeneous parts of the dissipation rate are

\[ \varepsilon = \bar{\varepsilon} + D \tag{2.7} \]

The eddy viscosity \( \mu_t \) is defined by:

\[ \mu_t = \rho C_f \frac{k^2}{\bar{\varepsilon}} \tag{2.8} \]

For high-Reynolds number flows the standard k-ε models constants are:

\[ C_f = 0.09, \sigma_k = 1, \sigma_\varepsilon = 1.3, C_{f1} = 1.4, C_{f2} = 1.92, f_P = 1, D = E = 0, \bar{\varepsilon} = \bar{\varepsilon} \]

As we mentioned before, the STD k-ε model has some limitations near solid boundaries, consequently, the wall functions approach or a well-defined damping function is used in order to resolve the associated problems near solid boundaries. These assumptions could be valid for e.g. slowly-developing boundary layers; however, it would be dodgy in highly non-equilibrium regions (particularly near impingement, separation or reattachment points).

The \( \nu^2-f \) model was introduced to overcome the limitations of second moment closure models. It is able to predict accurately the damping of turbulence transport near solid boundaries without using either wall functions or damping functions. In addition, it can reproduce the well-known near wall non-local effects of pressure deformation fluctuations. The \( \nu^2-f \) model including its variants has been validated over a wide range of complex flows.

The \( \nu^2 \) transport equation (regarded simply as a velocity scale that satisfies the boundary conditions appropriate for the normal component of turbulent intensity) can be described as:

\[
\frac{\partial}{\partial t} (\rho \nu^2) + \frac{\partial}{\partial x_j} (\rho u_j \nu^2) = \frac{\partial}{\partial x_j} \left[ \mu + \mu_t \frac{\nu^2}{\bar{\varepsilon}} \right] + \rho (k f - \nu^2 \frac{k}{\bar{\varepsilon}}) \tag{2.9}
\]

The quantity \( \nu^2 \) is obtained from a transport equation simplified from second-moment closure. The associated pressure strain term responsible for redistribution of turbulence energy in proximity of walls in order to return the correct level of turbulence anisotropy is obtained by solving an auxiliary elliptic relaxation equation for \( f \) :

\[
L' \frac{\partial^2 f}{\partial x_j^2} - f = 1 \left[ C_i - 6 \nu^2 k - \frac{2}{3} (C_i - 1) \right] - C_{fr} \frac{\rho}{k} \tag{2.10}
\]

The turbulent viscosity \( \mu_t \) now is defined as:
\[ \mu = \rho C_{\mu} v^2 T \]  \hspace{1cm} (2.11)

The turbulent time \( T \) and length scale \( L \) are given by:

\[ T = \max \left( \frac{k}{\varepsilon}, \frac{V_c}{C_{15}} \right) \rho \]  \hspace{1cm} (2.12)

\[ L = C_L \max \left( \frac{k^{1.5}}{\varepsilon} \rho^{0.75}, C_y \rho^{0.25} \right) \]  \hspace{1cm} (2.13)

The constants of the friendly model\(^{[21]}\) are

\[ C_\rho = 0.22, \quad \sigma_1 = 1, \quad \sigma_2 = 1.3 \]

\[ C_{\sigma_1} = 1.4 (1 + 0.05 \sqrt{\frac{k}{\varepsilon}}), \quad C_{\sigma_2} = 1.9 \]

\[ C_i = 1.4, \quad C_3 = 0.3, \quad C_4 = 0.23, \quad C_5 = 70. \]

The nonlinear \( k-\varepsilon \) turbulence model is obtained by using either the quadratic or cubic form of the strain-strain relationship as proposed by\(^{[20]}\):

\[ u_i u_i = \frac{2}{5} k \delta_{ij} - 2 C_{\varepsilon} f \frac{k^2}{\varepsilon} \eta \]

\[ \frac{\varepsilon}{k} \left[ C_3 (S_u S_u - \frac{1}{3} S_{i j} \delta_{ij}) + C_4 \left( S_u \Omega_u + S_u \Omega_u \right) \right] + \]

\[ \frac{\varepsilon}{k} \left[ C_5 (S_u \Omega_u + S_u \Omega_u \Omega_u) \right] + \]

\[ \frac{\varepsilon}{k} \left[ C_6 S_u + C_7 \Omega_u S_u \right] + \]

where the main strain and vorticity tensors are defined by:

\[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right), \quad \Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \]  \hspace{1cm} (2.16)

The model constants are given by:

\[ C_\varepsilon = \frac{0.3}{1 + 0.35 \rho^{0.15} \exp(-0.6 \exp(0.75 \rho))} \]

\[ \eta = \frac{\max(\varepsilon, \Omega)}{\varepsilon} \]  \hspace{1cm} (2.17)

\[ \Omega = (k/\varepsilon) \frac{2 \Omega_u}{S_u} \]

The dimensionless parameters \( \beta, \Omega \) are defined as the normalized strain and vorticity tensor. The different coefficients are given by:

\[ C_i = -0.4 C_{\mu}, \quad C_{\varepsilon} = -C_i, \quad C_5 = 1.04 C_{\mu}, \quad C_6 = 80 C_{\varepsilon}, \quad C_7 = -0.4 C_{\mu}, \quad C_b = -C_5 \] for cubic model, while

\[ C_a = C_b = C_\varepsilon = 0 \] for quadratic model\(^{[20]}\).

**Numerical procedure**

The previous governing equations are discretized, integrated and solved using the control volume approach on a non-staggered computational grid. All the dependent variables of interest here follow the generalization of conservation principle. If the dependent variable is denoted by \( \varphi \), the general differential equation for steady, incompressible, two-dimensional \( (x = 0, r = y) \) or axisymmetric \( (x = 1) \) is given by:

\[ \frac{\partial}{\partial t} (\rho \varphi) + \frac{\partial}{\partial x} (\rho u \varphi) + \frac{\partial}{\partial r} (\rho v \varphi) = \frac{1}{r} \left( \frac{\partial}{\partial r} (\Gamma \varphi r^2 \frac{\partial \varphi}{\partial r}) + \frac{\partial}{\partial r} (\Gamma \varphi r^2 \frac{\partial \varphi}{\partial r}) + S_\varphi \right) \]  \hspace{1cm} (2.18)

where \( \varphi \) is the dependent variable, \( \Gamma_\varphi \) is the diffusion coefficient for \( \varphi \), and \( S_\varphi \) is the source term. The quantities \( \Gamma_\varphi \) and \( S_\varphi \) are specific to a particular meaning of \( \varphi \). The above general differential equation can be written in terms of the total fluxes over the control volume faces and the resulting equation is integrated over each control volume. In similar manner, the continuity equation is integrated over the control volume. By manipulating the two integrated equations, one can derive the discretization equations. For more details, the reader is referred to\(^{[23]}\).

The discretization equations relate the unknown value of dependent variable \( \varphi \) at time \( t + \Delta t \) located at the pole of the control volume to those neighboring points and the known values at time \( t \). The source term \( S_\varphi \) is linearized by splitting it into \( S_C \) which stands for the constant part of \( S_\varphi \), and \( S_b \) the coefficient of \( \varphi \).

**3. RESULTS AND DISCUSSIONS**

### 3.1 The self-preserving axisymmetric turbulent jet

Although the self-preserving axisymmetric turbulent jet is considered as a relatively simple turbulent shear flow, however, it represents a benchmark for research into the physics of turbulent fluid flow. The importance of the free jet to the understanding of turbulence is evidenced by the large number of publications involving experimental data,
mathematical analysis and computational modeling. Many relevant technical applications are passed upon jets, e.g. rocket engines and gas turbines.

It was established that the jet was truly self-preserving some distance downstream of the nozzle and therefore most of the results were presented in excess of this distance. However, the turbulent intensity show marked departures from self-preservation even on the axis of the jet. Therefore, the numerical simulation of the axisymmetric turbulent jet is presented in the present paper to investigate the performance of the implemented turbulence models. Great emphasis should be placed on the prediction of the Reynolds stresses that can describe the flawless performance of the used turbulence models. Indeed, jets pose a difficult problem for turbulence models employing a single fixed set of constants. Jets are therefore widely used as a standard test case in turbulence modeling evaluation.

The axisymmetric turbulent jet used in the numerical experiment reported here, as shown in Figure 1, was modified to closely match the boundary conditions. Air with a mean velocity of $u_{in}=51$ m/s exits through a central nozzle with a diameter of $D_n=26.4$ mm into ambient air at atmospheric pressure. In this case, the Reynolds number is $Re=10^5$ and the Mach number is $Ma=0.15$, both allowing the flow to be considered incompressible. It was concluded in [27] that the effect of the initial conditions near the nozzle exit diminishes rather slowly with downstream distance; therefore the computational domain is allowed to extend more than 70 diameters downstream the nozzle. At the nozzle exit a uniform velocity distribution is employed. The turbulent intensity at the nozzle exit is estimated using the turbulent fluctuating to mean velocity at nozzle exit by $Tu=\frac{u'}{u_{in}}=0.16$ $Re^{0.125}$. The resulting inlet kinetic energy is $k_{inlet}=1.5(u_{in}Tu)^2$ and the corresponding dissipation rate $\varepsilon_{inlet}=(C_v\frac{u'}{u_{in}})^{2/3}(k_{inlet})^{-1/3}/l$ where $l$ is the turbulent length scale, assumed to be the nozzle diameter. Due to the symmetry of the case considered, only the upper half of the computational domain is simulated. The non-equidistant numerical grid employed here consists of 101 (axial) by 151 (radial) grid cells for a computational domain of $3 \times 1.0$ m, respectively. Four types of boundary conditions were used to describe the flow field within the computational domain; namely: inlet boundary conditions at the nozzle exit, axis of symmetry along the jet centerline, outlet boundary conditions at the top and the right end of the computational domain, while slip boundary conditions are applied at the side-west wall.

The early experimental investigation of [26] has been repeated in [28] by using both stationary and flying hot-wire and burst-mode LDA techniques for the same source and boundary conditions. The results obtained in [28] differed substantially from those reported in [26]. These differences were attributed to the smaller enclosures used in the earlier experiment in [26] and the recirculation within them. Also, the flying hot-wire and burst-mode LDA measurements differed from the stationary wire measurements due the effect of the high turbulence intensity at the jet centerline on the performance of the measuring instruments. According to such associated experimental problems, besides the superior variability of simulations (in the choice of fluid parameters and of initial conditions), the numerical simulations becomes recently an important source of information that is not available experimentally.

Centerline velocity variation and mean velocity profile

Figures 2-3 show the predicted centerline velocity and the mean axial velocity distribution for the axisymmetric turbulent jet, predicted by the implemented turbulence models, and compared with the experimental measurements of [26,28]. The experimental measurements of [26] were carried out by using the linearized constant-temperature hot-wire anemometers; while the experimental measurements of [28], have been performed by using the stationary hot-wire (SHW) and the burst-mode laser-Doppler anemometry (LDA). Figure 2 illustrates the variation of the centerline velocity, normalized by the inlet mean velocity, as a function of the axial location normalized using the nozzle diameter. Figure 3 shows the mean axial velocity profile in the radial direction, normalized by the centerline velocity and plotted versus the non-dimensional radial coordinates at an axial location of $x/D=75$.

Figure 2 shows the decay rate of the centerline velocity that should by a straight line revealing the self-preserving characteristics of the jet. The greater the slope of the line, the higher the decay rate of the centerline velocity. The virtual origin of the jet is represented by the initial nearly constant part of the profile. All the turbulence models adopted predict well the linear distribution of the centerline velocity. However, the results predicted by the $v^f$ model are the nearest results to the experimental measurements made by using the SHW and the prediction of the virtual origin of the jet, nearly $x=4.5D$, which is in close agreement with the prediction of [28]. The same conclusion is obtained in figure 3 for the mean velocity profile in the radial direction at $x/D=75$. The $v^f$ model predicts the best results compared with the experimental data using the SHW. The differences between the experimental measurements of [26] and those of [28] are attributed to the experimental facilities, while the discrepancies between the two experimental techniques SHW and LDA are small but have non-negligible effects. Therefore, for the mean velocity, it is difficult to assess the performance of the turbulence models adopted.
Reynolds stresses (second moments of velocity)

The Reynolds stresses in both radial and tangential directions are non-dimensionalized by the square of the centerline velocity and compared, for all turbulence models adopted, with the experimental measurements of [28], as shown in Figures 4, 5. The axial component of the turbulence kinetic energy is plotted in Figure 4. The STD $k-\varepsilon$ and the $v^2f$ models showed a good agreement with the LDA data than the nonlinear model, which overpredicts the experimental measurements. That can be attributed to the excessive prediction of the normal stresses by the nonlinear models in such cases where the anisotropy stresses effects have a less contribution to the Reynolds stresses. The prediction of the radial component of the turbulence kinetic energy is shown in Figure 5, where all turbulence models overpredicted the experimental measurements. That might be attributed to the effect of the leading cross-flow error on the measuring instruments[28]. In general, the STD $k-\varepsilon$ and the $v^2f$ model showed nearly the same distribution. However, the nonlinear model showed a different profile.

The turbulent shear stress is plotted in Figure 6. All the turbulence models adopted in the present paper showed an overall qualitative agreement with LDA measurements, while the SHW measurements failed to predict the distribution of the turbulent shear stresses. A fairly quantitative agreement has been obtained for the STD $k-\varepsilon$ and $v^2f$ models. However, the nonlinear model still overpredicts the experimental measurements.

3.2 Impinging problem

More recently, impinging jets have received considerable attention because of their wide engineering and industrial applications. Examples include manufacturing, material processing, cooling of turbine blades and drying paper, textiles, quenching of metals and glass, and more recently cooling of electronic equipment[29,7]. There are numerous papers dealing with this problem both numerically and experimentally focusing essentially on the prediction.
of the heat transfer parameters\cite{30}. However, the prediction of the turbulent characteristics is less investigated as a result of the complexity associated with the experimental measurements or the numerical challenges.

For a better understanding of the jet impingement heat transfer process, details of the flow and turbulence characteristics are required\cite{31}. Consequently, the numerical simulation of impinging jet problem would have been an important key for quantifying the effect of different parameters of interest. However, many complex features are encountered in the impinging problem due to stagnation, jet entrainment and high streamline curvature. These features prove to be incompatible with most existing turbulence models, which are essentially developed and tested for flows parallel to a wall. Therefore, the complexity of this flow has led it to being chosen as an excellent and challenging test-case for the validation of different turbulence models. The geometry of the impinging jet is shown in Figure 7, in which, a turbulent air flow issued from a nozzle, with diameter D and uniform exit velocity U. The problem is governed by physical, hydrodynamic and geometrical parameters of the system jet/surface. The principal parameter, in addition to D and U, is the distance H between the nozzle and the surface of impact, generally expressed in the dimensionless form H/D.

![Figure 4 Axial component of kinetic energy](image1)

![Figure 5 Radial component of kinetic energy](image2)

![Figure 6 Turbulent shear stress for axial jet](image3)
The boundary conditions are required to be specified on all surfaces of the computational domain. At the axis of symmetry, the radial velocity component and the gradient of the other dependent variables were equal to zero. Along the solid wall, the no-slip boundary condition is applied for velocity components, and a zero value is assigned for the turbulent kinetic energy and the gradients of energy dissipation rate. The inlet boundary conditions imposed at the nozzle are described as in the case of turbulent jet (cf. section 3.1). The computational grid consists of 150x80 uniform grid points in y and r directions, respectively. Two values of nozzle to plate distance (H/D=2 and H/D=6) have been considered for Reynolds number of 23000. The experimental case selected to validate the results of this work is a set of flow measurements carried out by [32].

**Axial velocity profiles along the jet axis**

The performance of the three-selected turbulence models is considered in this section for the impinging jet problem. The axial velocity profiles on the stagnation line is shown in Figure 8 for the two nozzle to wall distances (H/D=2, H/D=6) and compared with the experimental measurements of [32]. The calculated profiles were normalized by the bulk velocity (average velocity over the nozzle cross sectional area). Figure 8 showed that there is very little difference between the three predictions and quite a good agreement with the experimental data is observed for the two ratios of H/D considered. Therefore, generally, it is difficult to assess the performance of the turbulence models by presenting the mean velocity profiles. Near the solid boundaries, the implemented wall functions for the STD k-ε model give almost the same results of v²/ε model and the nonlinear k-ε model without using any wall functions.

**Axial Reynolds stress component**

In the present section, a sample of the obtained numerical results is only presented to show the performance of the turbulence models adopted. Figure 9 illustrates the development of the axial Reynolds stress profile as the flow develops away from the stagnation point for the two nozzle to wall distances (H/D=2, H/D=6) and for Re=23000. The comparison of the numerical results obtained showed that none of the implemented turbulence models perfectly matches the experimental data. However, the nonlinear k-ε model and the v²/ε give much reliable results than the STD k-ε. The comparison of the Reynolds stresses components at different locations of r/D give also the same behavior (not shown here). Nevertheless, the mean velocity profiles predicted by all turbulence models are reasonable and good. The v²/ε turbulence model shows quite reasonable results although the Boussinesq concept is applied to compute the Reynolds stresses. The nonlinear model is expected to perform consistently better than other linear models as a result of the cubic relation coupling the strain with the vorticity and stress tensors. However, the quantitative comparisons showed very little differences. These results have been also obtained in the original paper of [20], where little differences were observed between the experimental measurements [32] and the profiles of rms velocities perpendicular and parallel to the wall. We can refer that to the choice of the model coefficients C₁,…,C₇ and the expression
Ashraf Balabel and Wageeh El Askary

proposed for $C_\mu$ and $f_\mu$. This is a general problem of either linear or nonlinear turbulence models.

It is essential to check the range of validity of the different turbulence models because of most of the to-date-turbulence models are sometimes fitted for a given test-case, and might give much worse results when flow conditions are changed. Therefore, the dependence on the nozzle-to-plate distance is shown in Figure 9.

**Turbulent viscosity structure**

In order to show the effectiveness of the selected group of turbulence models, the structure of the turbulent viscosity is illustrated in Figure 10 for the nozzle to wall distance $H/D=6$ and $Re=23000$. The STD $k-\varepsilon$ model shows an excessive generation of the turbulent viscosity near the stagnation point. This can be attributed to the misrepresentation of the turbulent kinetic energy production rate. This problem is much less observed when using the $v^2f$ model and almost disappears by using the nonlinear representation of the Reynolds stresses components.

An important feature of the turbulent viscosity structure is shown in Figure 11 by comparing the results of the nonlinear $k-\varepsilon$ model in case of $H/D=2$ and $H/D=6$ for $Re=23000$. For smaller nozzle to wall distance, e.g. $H/D=2$, the evolution of the issued jet is not complete. The incoming jet hits the wall in a wide area than that observed for $H/D=6$ and the reflecting flow is characterized by a highly disturbed eddy structure on both sides of the impinging jet. This makes most of the experimental measurements in that region much complex and less accurate. However, for larger nozzle to wall distance, $H/D=6$, the incoming jet is considered to be fully expanded regarding to the trajectory of the jet due to the larger impinging distance compared with the initial mixing region required, that can extend to an $x/D$ of about $S^{(3)}$. Consequently, the impinging flow moves nearly parallel to the wall.

It should be pointed out that, the highly disturbed eddy structure obtained for $H/D=2$ by using the nonlinear $k-\varepsilon$ model could not be predicted using the other linear models (e.g. STD $k-\varepsilon$ and $v^2f$ models), as illustrated in Figure 12. The success of the nonlinear $k-\varepsilon$ model in predicting the turbulent viscosity structure better than those models based on Boussinesq concept reveals that the nonlinear models are more suitable for predicting the impinging problem. However, the group of coefficient implemented in such models need to be validated over a wide range of numerical simulations as well as experimental measurements in similar applications in order to obtain good quantitative results. This problem will be considered in more details in our future research.
Implementation and Validation of Different k-ε Turbulence Models in Engineering Application

Figure 9 Axial Reynolds stress profiles for impinging jet problem at H/D=2 (left) and H/D=6 (right), (r/D=0), Re=23000

Figure 10 The turbulent viscosity structure for impinging jet problem at H/D=6 and Re=23000

Figure 11 The turbulent viscosity structure for impinging jet problem obtained by the nonlinear turbulence model at H/D=2 (left) and H/D=6 (right), Re=23000
3.3 Axisymmetric sudden expansion flow

In the present section, the numerical method developed here along with the selected turbulence models are applied further to predict the axisymmetric sudden expansion flows. These types of turbulent flows have been widely investigated either numerically or experimentally due to the complex nature of the flow encountered (e.g. the flow is unstable with a large-scale coherent structure).

The geometry shown in Figure 13 consists of two coaxial pipes with a smooth surface and with small diameter d=50 mm and large diameter D=95.2 mm, to achieve an expansion ratio of 1.904. The test pipe is 660.8 mm long and the Reynolds number based upon the inlet diameter (d) and the mean velocity is 84000. In such configuration, the fully turbulent flow comes from the upstream of the sudden expansion forming a thin boundary layer along the wall of the smaller pipe. When the pipe suddenly expands, the pressure gradient causes the new mixing layer to curve toward the wall of the larger pipe and bifurcate at the reattachment point. The flow undergoes rapid distortion in the region surrounding the reattachment point and becomes very complicated and embodies a wide variety of complex turbulent flow and then it subsequently relaxes after the reattachment point. Consequently, this test case has played a central role in benchmarking the performance of both the numerical method and the turbulence models for separated flows.

If the turbulence model can reproduce the encountered flow feature and the reattachment point correctly, then the possibility of the high performance of the model in other types of turbulent flow is greatly increased.

In order to assess the performance of the three models selected, the numerical results obtained in the present paper have been compared with the experimental measurements[34] conducted using LDV at a specified location of x/D=0.95 measured from the sudden expansion wall and located in the separated region.

The cell density is uniform along the x-axis and the cross sectional plane with a grid of 150x100 in x and r-directions, respectively. The velocity at the inlet is parallel to the tube axis without cross-stream velocity. Neumann boundary conditions are applied on the inlet and outlet sections revealing that the velocity components and turbulence quantities are assumed to be fully developed for the inflow, where the mass flux is specified, but to have a zero normal gradient for the outflow. The wall functions are only applied for the STD k-ε model at the side and upper walls of the tube. Symmetric boundary conditions are applied on the symmetric line while no-slip boundary conditions are applied on the channel upper wall.
Implementation and Validation of Different k-ε Turbulence Models in Engineering Application

Figure 14 The velocity profile (left) and the kinetic energy (right) for sudden expansion problem at x/D=0.95 and Re=84000.

Figure 14 shows the comparison between the predicted velocity profiles and the kinetic energy profile obtained from the three selected turbulence models in the separated region and compared with the experimental measurements\[^{34} \]. The axial velocity and the kinetic energy values are non-dimensionalized by the inlet velocity and the inlet kinetic energy. The velocity profiles of the three turbulence models show good agreement with the experimental measurements far away from the wall. However, near the wall, the prediction of the back flow is captured well only with the \(v^2f\) and the nonlinear k-ε models. Slight differences between the three models were observed in predicting such mean parameters. However, for the turbulent kinetic energy, the nonlinear k-ε model and the STD k-ε showed fair agreement with the experimental measurements. The surprising result is the well prediction of the kinetic energy profile obtained from the STD k-ε model than that obtained from the \(v^2f\) model. The large deviations of the \(v^2f\) model from the experimental measurements in the separated regions have also been observed in [18]. The reason for that is not clear and needs further investigations. It can be seen also that the nonlinear k-ε model overpredicts the maximum value of the experimental kinetic energy, while the STD k-ε model underpredicts the maximum measured values. These results can be attributed to the coefficients of the turbulence models adopted. A slight change in the coefficients values can lead to increase or decrease of the numerical-experimental deviations. This is approved by [35], and it is considered as the main general problem of the turbulence models.

An alternative measure for the turbulence models performance is the numerical prediction of the reattachment point compared with the experimental data. For sudden expansion turbulent flow, the experimental measurements showed that the reattachment point is located at value of x/D=1.97. The numerical results obtain here showed that this value equals 1.65, 2.04 and 2.05 for the STD k-ε, \(v^2f\) model and nonlinear k-ε model, respectively. This indicates the shortage of the STD k-ε model compared with advantages of the nonlinear models and \(v^2f\) turbulence models.

In general, the different predictions of the existed turbulence models in the well known engineering applications have motivated the researchers to develop a new group of turbulence models which is based essentially on extending the isotropic k-ε turbulence model accompanied by an elliptic relaxation approach to account for the distinct effects of the near-wall and low-Reynolds number\[^{36} \]. This developed group of turbulence models showed considerable improvement over the existed standard eddy viscosity turbulence models. This kind of research could provide the turbulence modeling researchers with new concepts and modeling facilities.

4. CONCLUSION

The performance of three selected k-ε based turbulence models have been investigated numerically. The selected turbulence models; the STD k-ε, \(v^2f\) and the cubic nonlinear k-ε models are validated against three axisymmetric flow problems; namely: free round jet, the impinging jet problem and the turbulent flow through sudden expansion.

The quantitative comparison of the predicted numerical results with the experimental measurements reveals that an excellent agreement has been obtained for the mean velocity profiles. However, none of the implemented turbulence models perfectly matches the experimental data when comparing the Reynolds stresses. These remarks have been also demonstrated
by the previous investigations of turbulence modeling. We refer that to the selected values/functions of the model coefficients and parameters.

The quantitative comparison with the experimental measurements for the mean parameters might tell us which model behaves well. However, that gives at the same time an ambiguous indication about the accurate prediction of the turbulent structure characteristics e.g. turbulent viscosity, Reynolds stresses contours and the prediction of the reattachment point in separated flows. It should be pointed out that, the prediction of the turbulent structure characteristics is much important than the prediction of the mean parameters. Therefore, the performance of the turbulence models can only be assessed based on good predictions of both kinds of results.

The problem of turbulence modeling is, to date, the turbulence models used for either free-shear flows or impinging problem. These models have not exhibit nearly the degree of generality as those used for wall boundary layers. It is still a major challenge to find models that can provide accurate predictions for the development of both the free and impinging jet without requiring adjustments in the model parameters. This subject should be the modern topic and the open problem in the further turbulence research.

REFERENCES


