Prediction of Turbulent Flow Characteristics in Asymmetric Sudden Expansion Using Linear and Non-Linear Turbulence Models

A. Balabel¹, ², *, W. A. El-Askary²

¹Mechanical Engineering Department, Faculty of Engineering, Taif University, Taif, Saudi Arabia
²Mechanical Power Engineering Department, Faculty of Engineering, Menoufiya University, Shebin El-Kom, Egypt

Abstract

Recently, various turbulence models are widely developed in order to present a good prediction of turbulent characteristics in a wide range of industrial and engineering applications. Since the linear eddy-viscosity models show a poor prediction for some modern applications and their assumptions become invalid for separated flows, the non-linear eddy viscosity models become an important topic in the recent modeling of turbulent separated flow. The non-linear eddy viscosity models are expected to give a better prediction of reattaching turbulent flow as a result of the non-linear constitutive stress-strain relationship proposed to extend the applicability of the linear-eddy viscosity models. In the present paper, different linear and non-linear turbulence models are applied to predict one of the most complex geometry turbulent flows; namely, turbulent flow in asymmetric divergent channel. The abilities of these models to predict the turbulent flow characteristics in such complex geometry are examined and investigated numerically by solving Reynolds-averaged Navier-Stokes equations using the finite volume approach. The computed results are compared with the available experimental data in literature. The comparison of the obtained results with the experimental data has showed that the non-linear models have a better prediction, for the separated region and the associated turbulent characteristics, than the linear models. Consequently, the non-linear turbulence models could be considered as a powerful and a reliable tool in predicting such complex turbulent flows.

Keywords

Asymmetric Sudden Expansion, Non-Linear Turbulence Models, Numerical Simulation, Separated Turbulent Flow

1. Introduction

Turbulence has a decisive influence on heat transfer, species transport, drag, vorticity distribution, separation and swirl flow. Separation and reattachment of turbulent shear layers in the presence of adverse pressure gradient can be seen in many practical industrial and engineering applications, either in internal flow systems such as diffusers, combustors and channels with sudden expansion, or in external flows like those past bluff structures and buildings. The major aerodynamic features are often dictated by a balance between pressure gradients and convection processes which are closely tied to geometric and through-flow constraints. When a flow contains significant regions of separation and recirculation, primary flow properties- notably the pressure field – are also affected, in which case turbulence influences even the qualitative behavior of the flow.

Previously, the primary approaches for studying turbulent flows was experimental and theoretical analysis. The time-averaged parameters are relatively easy to measure, however, some types of measurements, for example, the fluctuating pressure within a flow, are almost impossible to make at the present time. Others cannot be made with the required
precision. As a result, numerical methods have an important role to play [1]. Moreover, the rapid advance of computational schemes able, in principle, to analyse fluid flow and convective heat or mass transport over domains of arbitrary complexity again focuses attention on the method of characterizing turbulent exchange processes. Indeed, it is often said that turbulence is the pacing item for the rate of progress of CFD in fluids engineering, that reflected by the enormous amount of research on turbulence modeling over the past three decades. Efforts have focused mainly on the construction of two-equation eddy-viscosity models and Reynolds-stress-transport closure.

The importance of turbulence to at least some flow properties of major engineering interest poses serious challenges to the ability of computational approaches based on the solution of Reynolds-averaged Navier Stokes equations (RANS) to give quantitatively reliable predictions for any but relatively simple thin-shear flows. The RANS equations need a turbulence model for computation of Reynolds stresses that stems from averaging the non-linear convective terms, see for more details [2]. A large family of turbulence models exists in the literature. The models range from simple algebraic expressions for the eddy viscosity to more elaborate formulations which introduce a separate transport equation for each component of the Reynolds stresses, more detailed discussion can be found in [3] and [4].

Eddy viscosity models are based on Boussinesq assumption that relates the apparent turbulent shearing stresses to the rate of mean strain through an apparent scalar turbulent or “eddy” viscosity. Consequently, the relation between the Reynolds stresses and the velocity gradient is linear, and therefore, such models are termed linear eddy viscosity models. These models, e.g. standard k-ε model [5] and k-ω model [6] still represent a good compromise between accuracy and computational efficiency. Therefore, the two-equation eddy viscosity models have been the subject of much research in the last years as they still the most widely used in industrial and engineering applications, even though they fail to predict correctly a number of complex flows.

Some improvement of linear eddy viscosity models can be achieved through recalibrations of the standard k-ε and the k-ω models. This led many authors, for a further development of RANS model, to develop a modified form for the standard models, e.g. the renormalized group (RNG) k-ε model [7], the SST k-ω model [8], and the low-Reynolds-Number k-ε model [8]. Nevertheless, no pretence 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.

The alternative route thus pursued extensively has been the Reynolds-stress models which are not restricted by the Boussinesq approximation. These Models are often numerically challenging, computational expensive and, therefore, have some limitations in the context of industrial CFD. Moreover, the results of a recent workshop [9] showed that, even though full Reynolds stress models bring more physics into the model, the large increment in the computational effort associated with these models is not always followed by a proportional improvement in the quality of the predictions. At the present time, the Reynolds stress models have not even been tested for many types of complex flows, so this technique is much too costly at present to be considered as engineering tool.

The desire to combine the advantageous numerical properties and economy of Boussinesq-eddy viscosity models with the superior fundamental strength of Reynolds stress models which is mathematically complex and numerically challenging has motivated efforts to construct and formulate a new class of models with non-linear stress-strain relationships. These efforts have given rise to the group of non-linear eddy viscosity models.

In practical applications, the standard (STD) k-ε turbulence model is widely used owing to its simplicity and effectiveness. It is a robust two-equation model and it yields quite reasonable results in the case of high Reynolds number flow when its restrictions are taken into consideration [2]. It was developed, calibrated and validated for wall-bounded high-Reynolds number turbulent flows [3] and [5], and has been traditionally used in conjunction with empirical wall functions to patch the core region of the flow to the wall region. In some cases e.g., flow over the backward facing step or the high velocity spray, the STD k-ε yields a large turbulent diffusivity (viscosity) and thus an unrealistic flow behavior. Therefore, the numerical simulation of turbulent flows modeled by the k-ε model has been the subject of much research in the literature.

In order to overcome the poor predictions of the STD k-ε in some applications, a large number of modified k-ε models is treated as a RANS model and its applicability in simulating many industrial and engineering applications is investigated. The modification is performed by either re-evaluating the model constants [7] or by adding new terms to the epsilon equation [11], or through a modification of the production of the turbulent kinetic energy [12]. Some of such modifications have been succeeded in predicting a more realistic behavior of the flow in the high shear stress area; however, in the low shear stress area unrealistic results have been obtained [13].

Another alternative is to consider the non-linear turbulence
stress relationships. Non-linear eddy-viscosity models (NLEVM) can be traced back to the work of [14]. Such approach is generally found to marginally improve predicted turbulence intensities. However, relative to the linear models, convergence is mostly difficult to achieve. More details will be followed. Most non-linear eddy-viscosity models utilize constitutive equations which are functions of two turbulence scales (usually k and ε) as well as strain and vorticity invariants. These models can successfully model a wide range of flows in order to assess the ability of the models to predict anisotropy and streamline curvature effects, e.g. plane and fully-developed curved channel flow [15] and turbulent flow around a circular cylinder [16].

However, the non-linear models were not widely applied for reattachment of separated turbulent flows in the presence of strong adverse pressure gradients. In such situations, the flow exhibits the combined effects of three-dimensionality, streamline curvature, and streamwise pressure gradients. Among the flow geometries used for studies of separated flows, the most frequently selected is the backward-facing step due to its geometrical simplicity. In the backward-facing step flow, both an adverse pressure gradient and flow separation are to be modeled, which makes this test case particularly challenging. Moreover, the backward-facing step configuration is chosen as a benchmark case since it mimics the flow field in many engineering and industrial applications and existing numerical and experimental data are available for comparison.

In the present study, the performance of several turbulence models variants when these are applied for turbulent flow over a backward-facing step in asymmetric divergent channel has been investigated and compared with the available experimental data. The turbulence models adopted here are selected from a large family of turbulence models exist in the literature, mainly: Launder-Sharma (LS) model [17], Craft-Launder-Suga (CLS) model [18], and Lien-Chen-Leschziner (LCL) model [19]. The numerical results obtained from all investigated turbulence models will be verified via comparisons with the experimental data of [20].

2. Mathematical Model

We consider here the steady state turbulent flow of a viscous, incompressible fluid with constant properties and not influenced by gravity effects. The governing field equations are the Navier-Stokes and continuity equations, which are given by:

$$\frac{\partial}{\partial x_j} u_i = 0$$  \hspace{1cm} (2)

Where, $u_i$ is the velocity vector, $p$ is the pressure, and $\nu$ is the kinematic viscosity of the fluid.

Applying Reynolds averaging to the above equations yields a set of Reynolds-averaged Navier-Stokes equations and the time averaged continuity equation which are given, in Cartesian tensor notation, as

$$\frac{\partial}{\partial x_j} \bar{u}_i = -\frac{1}{\rho} \frac{\partial}{\partial x_i} p + \frac{\partial}{\partial x_j} (\nu \bar{S}_{ij} - \bar{u}_i u_j)$$  \hspace{1cm} (3)

and

$$\frac{\partial}{\partial x_j} \bar{S}_{ij} = 0$$  \hspace{1cm} (4)

where $S_{ij}$ is the main strain which can be represented as:

$$S_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)$$  \hspace{1cm} (5)

and $\bar{u}_i u_i$ is the Reynolds-stress tensor and $u_i$ represents the velocity fluctuation in $i$-direction. In these equations, the bar and prime indicate the time-averaged mean and the fluctuation from its mean value, respectively. These equations are not a closed set and turbulence models are required to model the turbulent or Reynolds-stress tensor.

2.1. Turbulence Modeling

The turbulence models employed for the present simulation are the standard linear $k-\varepsilon$ model and the recently developed different non-linear $k-\varepsilon$ turbulence models. Here we will discuss in brief the fundamental principles between linear and non-linear turbulence models.

2.1.1. Linear-Eddy-Viscosity Model (LEVM)

In the linear eddy viscosity turbulence model (LEVM), the Reynolds stress tensor is computed from the effective viscosity formulation, which is a direct extension of the laminar deformation law. It is given by

$$\tau_{ij} = \frac{2}{3} k \delta_{ij} - 2 \nu \bar{S}_{ij}$$  \hspace{1cm} (6)

where, $k = 0.5 \bar{u}_i u_i$ is the turbulent kinetic energy, $\delta_{ij}$ is the Kronecker delta and $\nu$ denotes turbulent kinematic viscosity which, unlike its laminar counterpart, varies spatially, and is not a property of the fluid. The eddy viscosity represents the product of two turbulent quantities, namely the length scale and velocity scale of the large energy-bearing eddies. The
choice of the second scale is not unique and different researchers have selected different quantities. In the present work, the turbulent eddy viscosity is defined as:

\[ v_t = C_f \mu k^2 \left( \frac{x}{\varepsilon} \right)^2 \]  

(7)

In order to obtain \( v_t \) and its distribution, two transport equations for the turbulence kinetic energy \( k \) and the homogeneous dissipation rate \( \varepsilon \) are solved. The most known popular turbulence model for solving engineering problems is the standard-linear \( k-\varepsilon \) model with the wall function, which was introduced by [5]. The \( k \) equation reads

\[ \frac{\partial}{\partial x_j} \left( \frac{\partial \mu}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\nu + \nu_i}{\sigma} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \varepsilon \]  

(8)

where the turbulent production rate is

\[ P_k = 2v_t S_{ij} S_{ij} \]  

(9)

The transport equation for the kinetic energy dissipation is given in the form of

\[ \frac{\partial}{\partial x_j} \left( \frac{\partial \varepsilon}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\nu + \nu_i}{\sigma} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \left( c_{1\varepsilon} \frac{\varepsilon}{k} P - c_{2\varepsilon} \frac{\varepsilon^2}{k} \right) \]  

(10)

The model coefficients \( c_{1\varepsilon} \), \( c_{2\varepsilon} \), \( \sigma_k \), \( \sigma_\varepsilon \) and \( C_f \) are given in Table 1 for the various models adopted. The variable \( \varepsilon \) is the homogeneous dissipation rate which can be related to the real dissipation rate through

\[ \varepsilon = \varepsilon + \Delta \]  

(11)

The dimensionless parameters \( \overline{k}, \overline{\varepsilon} \) are known as the normalized strain and vorticity tensor.

In the standard or high-Reynolds number \( k-\varepsilon \) models \( f_1 = f_2 = f_3 = 1 \), \( \Delta = E = 0 \) (diffusion terms) and \( \varepsilon = \varepsilon \).

The weakness of the linear-eddy-viscosity turbulence model also includes the inability to capture normal stress anisotropy and insensitivity to streamline curvature. Second-moment closure models, on the other hand, account for several of the key features of turbulence that are misrepresented by the linear turbulence models. However, these models are considerably more complex and require higher CPU time than \( k-\varepsilon \) model.

A simpler alternative for approximating of the Reynolds stress is to extend the strain-stress relation of the linear eddy-viscosity model, by adding all the higher order nonlinear combinations of the strain and vorticity rate tensors that satisfy the kinematic constraints of the turbulence stress tensor. The coefficients of these terms are then determined with reference to a range of basic flows. These nonlinear strain-stress relations have the ability to produce the differences between the normal stresses and thus can extend the model’s applicability. They can be used to predict flows in which the anisotropy of turbulence is important, such as flows involving turbulence-driven secondary flows (e.g. Dean cells in curved ducts or separation due to sudden divergence presented in our contribution). Although the ideas of NLEVM themselves emerged in the 70’S, until recently, the models of this type were not widely explored. Many attempts at developing and using such schemes have been recently made.

One of the advantages of the NLEVM is its similarity to the
standard k-ε model. It basically constitutes a two-equation model with an anisotropic eddy viscosity. The two equations are, as before, the transport equation of the turbulence kinetic energy \( k \) and dissipation rate \( \varepsilon \).

Most NLEVM are quadratic or cubic according to the order of the characteristics time scale \( \tau \) in the anisotropy tensor equation. These differences are in order of considerable significance. In particular, the cubic models play an essential role in capturing the strong effects of curvature on the Reynolds stresses.

In quadratic and cubic eddy-viscosity models, the stress-strain relationship can be written in the following form [15]:

The nonlinear components of Reynolds stresses are included in the RANS momentum equations as source terms. The various constants \( C_1 - C_7 \) are given in Table 2 according to the turbulence model adopted.  

\[
\tau_y = \frac{2}{3} k \delta_y - 2C_{f \mu} k^2 \mu \frac{k^2}{\varepsilon} S_{ij} + C_4 \left( S_{ij} S_{ij} - \frac{2}{3} \delta_{ij} S_{kk} \right) - C_3 \left( S_{ij} \Omega_{ij} + S_{ij} \Omega_{ij} \right) + C_5 \left( \Omega_{ij} S_{ij} S_{ij} + \Omega_{ij} \Omega_{ij} S_{ij} - \frac{2}{3} \Omega_{ij} \Omega_{ij} \Omega_{ij} \delta_{ij} \right) + C_7 \left( \Omega_{ij} \Omega_{ij} S_{ij} \right) \]

Table 2. Constants for the stress-strain relationship. LS: Launder-Sharma [17], CLS: Craft-Launder-Suga [18], LCL: Lien-Chen-Leschziner [19].

<table>
<thead>
<tr>
<th>Model</th>
<th>( C_1 )</th>
<th>( C_2 )</th>
<th>( C_3 )</th>
<th>( C_4 )</th>
<th>( C_5 )</th>
<th>( C_6 )</th>
<th>( C_7 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>LS</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>CLS</td>
<td>(-0.4C_{f \mu}f_{\mu} )</td>
<td>(0.4C_{f \mu}f_{\mu} )</td>
<td>(1.04C_{f \mu}f_{\mu} )</td>
<td>(80C_{f \mu}f_{\mu} )</td>
<td>0</td>
<td>(-40C_{f \mu}f_{\mu} )</td>
<td>(-40C_{f \mu}f_{\mu} )</td>
</tr>
<tr>
<td>LCL</td>
<td>(\frac{3f_{\mu}}{1000 + 5^4} )</td>
<td>(\frac{15f_{\mu}}{1000 + 5^2} )</td>
<td>(\frac{-19f_{\mu}}{1000 + 5^3} )</td>
<td>(80C_{f \mu}f_{\mu} )</td>
<td>0</td>
<td>(16C_{f \mu}f_{\mu} )</td>
<td>(16C_{f \mu}f_{\mu} )</td>
</tr>
</tbody>
</table>

### 2.2. Near-Wall Effects

The treatment of wall boundary conditions requires particular attention in turbulence modeling. Viscous stresses in the flow remote from a wall boundary are in general negligible in comparison to the turbulent stresses. However, as the wall is approached (low-Re region) the turbulent shear stress is damped and the viscous stresses become more important. This results in sharp gradients in the velocity, Reynolds stresses and other modeled quantities such as \( k \) and \( \varepsilon \).

#### 2.2.1. Low-Re Region for LEVM

One of the most widespread approaches of modeling near wall flows in the linear turbulence modeling is to use wall functions, which have two obvious advantages. They allow to save computing resources and to take into account the influence of various parameters, in particular, roughness by means of empirical correction. In the wall function, often the buffer layer is neglected and considered in the viscous sub-layer.

The incorporated zone lies in the range of \( 0 < y^+ \leq 11.63 \). The linear dependence of speed flow from wall distance reads:

\[
U^+ = y^+ \quad (13)
\]

where, \( U^+ = \frac{U}{U_e} \), \( y^+ = \frac{U_e y}{\nu} \) and \( U_e = \sqrt{\tau_w / \rho} \) is the shear velocity and \( \tau_w \) is wall shear stress.

In the logarithmic layer, (for \( y^+ > 11.63 \)), the Reynolds stresses exceed much viscous effects and the structure of velocity can be expressed in the form of the logarithmic law as

\[
U^+ = \frac{1}{\kappa} \ln \left( E y^+ \right) \quad (14)
\]

where \( E = 9.8 \), for smooth walls as assumed in the present work and \( \kappa = 0.41 \) is the Van-Karman’s constant. At the wall, the boundary value for the dissipation rate at the first near-wall-point, (identified by the subscript \( p \)), can be expressed as

\[
\varepsilon_p = \frac{C_{\varepsilon}^2 k_p^{1.5}}{\kappa y_p} \quad (15)
\]

The near-wall value of the turbulence kinetic energy \( k \) is
computed by solving the complete transport equation for \( k \) in the near wall control volume, with the wall shear stress included in the production term and zero normal gradients assumed for \( k \) at the wall.

## 2.2.2. Low-Re Region for NLEVM

For authors known the standard \( k-\varepsilon \) with its standard wall function fails to predict sufficiently the separated flow behavior, such as the separation location and the reattachment position as well as the pressure distribution in the presence of strong pressure gradient. For this purpose it is preferred to use the damping functions depending on near-wall turbulence Reynolds number and including terms describing the molecular transfer in the boundary layer zone.

Table 3 summarizes the damping functions, closure coefficients and treatment of dissipation rate at the wall (low Reynolds-number) flows. The introduction of the additional terms \( D, \ E \) and damping functions in the equations for \( k-\varepsilon \) need some more exact definitions in the field of low value of Reynolds number and in the immediate proximity of a wall.

<table>
<thead>
<tr>
<th>Model</th>
<th>( f_1 )</th>
<th>( f_2 )</th>
<th>( D )</th>
<th>( E )</th>
</tr>
</thead>
<tbody>
<tr>
<td>LS</td>
<td>( \exp \left( \frac{-3.4}{1+0.02R_y^2} \right) )</td>
<td>1</td>
<td>1-0.3exp(-R_y^2)</td>
<td>2ν(\frac{\partial U}{\partial y})^2</td>
</tr>
<tr>
<td>CLS</td>
<td>( 1-\exp \left( -\frac{R_y}{90} \left( \frac{R_y}{400} \right)^2 \right) )</td>
<td>1</td>
<td>1-0.3exp(-R_y^2)</td>
<td>2ν(\frac{\partial U}{\partial y})^2</td>
</tr>
<tr>
<td>LCL</td>
<td>( 1-\exp \left( -0.0198y^2 \right) \left( 1+2.5y/C_y \right)^{-2} )</td>
<td>1</td>
<td>1-0.3exp(-R_y^2)</td>
<td>0</td>
</tr>
</tbody>
</table>

\( R_y = k^2 / \nu, \ y = y / \sqrt{V / \nu}, \ y_p \) is the normal distance from the wall.

\( YAP \equiv \max(0.8(\gamma-1)y^2(2\ell / k)^{0.4},0), \ y = k^{0.4} / \epsilon_y, \ c_2 = 2.5, \ \epsilon^2 = (k^{1.5} / \epsilon) (C_{\mu}^{0.75} + 2k / y^2) \).

### 3. Boundary Conditions

The computation domain and boundary conditions are set up as shown in Fig.1. The lower and upper boundaries correspond to the wind tunnel walls of Driver and Seegmiller’s experiment [20]. No-slip boundary conditions are imposed along the walls. The inlet velocity profile is assumed for \( 1/7 \)-th-power law. The inflow turbulence kinetic energy and its dissipation rate appropriate for the turbulent boundary layer are estimated by assuming local equilibrium of turbulence within the boundary layer [21];

\[
k = \ell_w^2 \left( \frac{\partial U}{\partial y} \right)^2 / C_{\mu}^{0.2} \quad (16)
\]

\[
\varepsilon = k^{1.5} C_{\mu}^{0.75} / \ell_w \quad (17)
\]

A ramp function distribution of the mixing length \( \ell_w \) is assumed. That is,

\[
\ell_w = \kappa y \quad \text{for} \quad y \leq C_{\mu} \delta / \kappa
\]

\[
\ell_w = C_{\mu} \delta \quad \text{for} \quad y > C_{\mu} \delta / \kappa
\]

The freestream value of \( k \) and \( \varepsilon \) at the inflow are estimated from the following relations:

\[
k_{in} = 0.0001U_{in}^2 \quad (19)
\]

And

\[
\varepsilon_{in} = k_{in}^{1.5} / C_{\mu}^{0.25} \delta \quad (20)
\]

The normal gradients at the outlet plane are taken to be zero except the streamwise velocity, which must be corrected every iteration step to satisfy the continuity equation.

### 4. Numerical Procedure

The governing Reynolds-averaged equations for the turbulent flow in asymmetric divergent channel, shown in Fig. (1-a), can be written for steady, incompressible, two-dimensional flows in the following generic transport equation form:

\[
\nabla \cdot ( \rho \mu \phi ) = - \nabla p + \nabla \left[ \Gamma_{\phi} \nabla \phi \right] + S_{\phi}
\]

where the variable \( \phi \) is the dependent variable, representing the streamwise velocity \( U \), the normal velocity \( V \), the turbulent kinetic energy and its dissipation rate \( k, \varepsilon \), respectively. The diffusion coefficient \( \Gamma_{\phi} \) and source term \( S_{\phi} \) in the respective governing equation are specific to a particular meaning of \( \phi \), see for more details [22].
The numerical method employed here to solve the above general differential equation is based on a general method for prediction of heat and mass transfer, fluid flow, and related processes. This method has been developed and proved its generality and capability in a wide range of possible applications for predicting physically meaningful solutions even for coarse grid [22]. The control volume integration of the above general differential equation (21), yields a discretized form being solved numerically on a staggered grid system shown in Fig. (1-b). The governing equations were discretized using the second-order upwind scheme to achieve the best accuracy. In this paper the SIMPLE algorithm is employed. The algorithm is started with the solution of the discretized momentum equations according to the associated boundary and initial conditions. A pressure correction equation, derived from the integration of the continuity equation, is then solved and the solution is used to update the guessed pressure and velocity fields. The all other discretized transport equations are then solved. The flow diagram is further iterated until the convergence is achieved in the order of $10^{-4}$ in velocities and of $10^{-3}$ in pressure.

5. Results and Discussion

The turbulent flow past a backward-facing step has played a central role in benchmarking the performance of turbulence models for separated flows. This paper will also use this class of complex flows to evaluate the different turbulence models adopted. When a channel suddenly expands at a step, the pressure gradient causes the new mixing layer to be curved toward the wall and to bifurcate at the reattachment point. One branch develops as a new boundary layer after the reattachment point and the other branch forms the recirculation region. Therefore, the flow undergoes rapid distortion in the region surrounding the reattachment point and subsequently relaxes downstream of this point. As a result, the flow downstream of the backward-facing step becomes very complicated and embodies a wide variety of complex turbulent flows. If a turbulence model could produce this flow correctly, then the possibility that this model may be successful with other type of turbulent flow is greatly increased. The experimental data of Driver and Seegmiller [20] for a turbulent flow over a backward-facing step are used in this section for comparing the numerical results obtained by using the standard as well as the nonlinear turbulence $k-\varepsilon$ models.

The inflow and geometrical parameters are described as follows; $U_{in}=42.2 \text{ m/s}$, $H=12.7 \text{ mm}$, $Re_{H}=37500$, $\delta=19 \text{ mm}$ and $ER=(H_{in}+H)/H_{in}=1.125$, where Reynolds number $Re_{H}$ is based on the step height $H$, $U_{in}$ is the freestream velocity at the inflow, $\delta$ is the boundary layer thickness at $4H$ upstream of the step, $H_{in}$ is the channel height before the step and $ER$ is the channel expansion ratio. The computational domain extends $10H$ upstream and $40H$ downstream of the step. In all discussions and figures, the nonlinear turbulence models used are abbreviated to Launder’s model [17], Craft’s model [18] and Lien’s model [19]. Since the turbulent flow over a backward-facing step is characterized by the recirculation zone after backward step and its reattachment point, the specific objective of this simulation is the demonstration of the advantages of non-linear turbulence models in predicting the characteristics of the recirculation zone compared to the standard two equation models.

The reattachment length $X_r/H$ of the four models Standard model, Lien’s model, Launder’s model and Craft’s model are given in Table 4 compared to a value of about 6.26 in the experimental data of Ref. [20]. It can be noticed that, the Craft’s model with $73 \times 73$ grid reproduces the reattachment
length almost the best compared with the other turbulence models with the same grid resolution, whereas the finest grid of Craft’s model improves the result to be very close to the experimental result.

5.1. Validation of the Obtained Results

Figure 2 shows a comparison of computed and experimental skin friction distributions $C_f = \frac{\tau_w}{0.5 \rho U_w^2}$. The Craft’s model (CLS) performs significantly better than the other models, while Lien’s and Launder’s models predict significantly large variations of $C_f$ in the recirculation zone and some insignificant differences downstream the reattachment point. A lower skin friction coefficient has been obtained from the standard model after the reattachment point. Actually, in Fig. 2 of the skin friction prediction, it is noticed that the experimental data has a little jump in its value just downstream of the step. This phenomenon may be corresponds to the formation of a cornered eddy. The same behavior is observed in the present numerical data obtained from all turbulence models used.

The surface pressure coefficient is defined as:

$$C_p = \frac{P_w - P_{ref}}{0.5 \rho U_m^2}$$

where $P_w$ is the wall static pressure and the subscript $\text{ref}$ represents the pressure at a reference position. The reference position considered here is that of the experiment of Ref.[20], which is located at $x / H = -5.1$.

Figure 3 reflects the trends already seen in $C_f$ distribution. The smaller the separation region predicted by the model the smaller is the displacement effect of the boundary layer and the larger is the pressure rise in the expansion region after the step. The Craft’s model slightly underpredicts the separation length and therefore gives a somewhat smaller pressure increase than the other models. A significant deviation of the standard model compared with the other nonlinear turbulence models is also noticed after reattachment point.
To further investigate the quality of the present turbulence models, the mean streamwise velocity profiles of the measured data in Ref. [20] and the computational results at various positions downstream of the step ($x/H = 1, 2, 3, 5$ and 32) are compared in Fig. 4. It can be seen that the Craft’s model performs better in predicting the velocity profiles near wall in the recirculation region. This improvement may be due to the better introduction of anisotropic turbulence stresses in the model of Craft compared with the other turbulence models. One notices that the standard model introduces good prediction in near of the corner point ($x/H = 1.0$) at the near-wall region, but there is an overprediction with farther distance from the wall. The same result is observed near the reattachment point ($x/H = 5.0$) with much more overprediction in the outer layer. The sudden changes in flow properties around the corner and near the reattachment point are responsible for the failing of the nonlinear models in predicting exactly the near wall backflow peak. At the last station ($x/H = 32.0$), one notices that all nonlinear turbulence models predictions are in reasonably good agreement with each other and a clear overprediction of the standard model. This can be explained as; the standard model could not predict the growth of the separated shear layer, which delays its ability to follow the developing process of the boundary layer after separation.
The quality of the nonlinear turbulence models is further extended to the turbulent shear-stress results ($-\overline{uv}$), as seen in Fig. 5. The computations using the Craft’s model are quite good overall, but somewhat underprediction of the peak values at the same failing regions of the velocity profiles ($x/H = 1.0$ and $5.0$), where as discussed previously there are sudden changes of flow properties. The other nonlinear turbulence models show visible underpredictions at all given stations. Also, it can be observed the effect of non considered streamline curvature in the standard model, where there is a large tendency of the standard model to overpredict the shear stress compared with the experimental data of [20]. The same prediction with the standard model was discussed by Driver and Seegmiller [20].
5.2. Effects of Grid Resolution

Figures 6 to 8 show the effect of grid resolution on the computed turbulent stresses profiles using the Craft’s model with different grids, 73×73, 103×103 and 153×153. The other linear- and nonlinear-turbulence models show visible underpredictions (not shown here), so we consider only the results of Craft’s model at the main important locations, especially in recirculation zone. The normal stresses \( u' u' \) and \( v' v' \) are shown in Figs. 6 and 7 respectively, while the turbulent shear stress \( u' v' \) is presented in Fig.8. It is noticed that the refinement of grid plays an important role in predicting the near wall behavior of the normal stresses at all locations presented here. Some deviations are observed away from the wall. This improvement is developed only in the near-wall region to the shear layer which starts at the step and ends by the reattachment point and can be observed as peaks of the normal stress profiles. That reveals the possibility of the nonlinear turbulence models especially that of Craft’s model in improving the prediction of turbulent stresses with presence of strong streamline curvature (which produced here by the separation shear layer). The small deviation developed upper the shear layer locations (over the peaks) may be explained as the effects of turbulence models coefficients \( c_1 \) and \( c_2 \) which mainly depend on the grid resolutions. This is a general problem for either the linear or nonlinear \( k-\varepsilon \) turbulence models [23].

From Fig. 8, it can be noticed that, in the vicinity of wall at all locations with simulating the turbulent shear stress \( u' v' \), some underpredictions with further refinement of grid resolution. This may be explained as the more increased level of inhomogeneous part of dissipation term \( D \) with more reduction of grid distance, which appears in more reduction of length scales and in turn reduces the values of the turbulent shear stress near the wall. However, there are no important effects of grid refinement in the regions upper the shear layer edge, because of the disappearance of this inhomogeneous part of dissipation with farther distance from the wall.
6. Conclusion

Different linear- and nonlinear-turbulence models were discussed and tested in the present paper. Both models were applied to a selected documented work that is meaningful for one-wall duct expansion producing shear layer and generates complex flow after the wall step. The nonlinear turbulence models with the considered near-wall low Reynolds wall functions were introduced. The numerical results from the standard $k-\varepsilon$ turbulence model are also presented. The results of the computations were compared to each others as well as against available experimental data from previous literature.

The central part of the comparisons is for the behavior of the chosen turbulence models under adverse pressure gradient condition and the generated curved-shear layer accompanied with separated flow. Comparisons with the published experimental data showed that the nonlinear turbulence models give improved numerical results compared with the
standard turbulence model for this type of problems. The results confirmed the findings of the adverse pressure gradient computations. The Craft’s model predicted highly accurate velocity profiles, almost identical to those of the experiments. The other nonlinear turbulence models (Lien’s and Launder’s models) have a larger sensitivity to the adverse pressure gradient than the Craft’s model and therefore predicted more retarded profiles.

A very surprising result of the computations is that the standard turbulence model gives more accurate solutions than that of Lien’s and Launder’s nonlinear turbulence model in the near wall region of reversed flow zone. This leads to unmodified results when using Lien’s and Launder’s model in predicting the reattachment point. One noticeable difficulty with all models is the inability to locate the peak of turbulence Reynolds stresses which indicates the progress of the developed shear layer. This may be a result of inadequate grid resolution near the walls for these types of models. However, in all comparisons one finds that the best tested turbulence model is that of Craft. A further study was also introduced to investigate the grid resolution dependences of the results of Craft’s model. For this problem, the Craft’s model gives better results in a good prediction of reattachment length within the uncertainty of the experiments and gives better representation of the normal turbulence stresses. Some underpredictions of turbulent shear stress with more refinement of grid resolution of Craft’s model have been observed. The reduction of the predicted turbulent shear stress may be due the more predicted length scale of turbulence near the wall.

References


