ROBUST EDDY VISCOSITY TURBULENCE MODELING WITH ELLIPTIC RELAXATION FOR EXTERNAL BUILDING FLOW ANALYSIS

Mirza Popovac
AIT Austrian Institute of Technology
Österreichisches Forschungs- und Prüfzentrum Arsenal Ges.m.b.H.
Energy Department, Sustainable Building Technologies
Giefinggasse 2, 1210 Vienna, Austria

ABSTRACT
This paper presents the application of the modified $\zeta-f$ turbulence model for the CFD analysis of the external building flows. The proposed modification, based on the elliptic relaxation eddy viscosity concept, enables very easy model implementation into a general purpose RANS-based CFD code, while keeping all advantages of the elliptic relaxation turbulence modeling approach: better physical description of the near-wall flow effects, and thus improved accuracy and reliability of the wall-bounded flow predictions. High robustness of the presented model modification, as well as its straightforward implementation (for the work in the present paper the model implementation in FLUENT is performed), makes it suitable for numerical analysis of the complex wall-bounded fluid flow and heat transfer engineering problems. In this paper the accent is on the external building flows, so the performance of this model and its implementation are tested on the realistic case of a flow around buildings in urban area. The obtained results enable the analysis of the flow effects that are characteristic to the external building flows, as the quality of the results is satisfactory, and the computation stability using the presented model was very good.

INTRODUCTION
The Computational Fluid Dynamics (CFD) became a very powerful tool for the analysis of complex fluid flow engineering problems. It incorporates modeling of various flow effects, such as turbulence, heat and mass transfer. Concerning the turbulence models, however, vast majority of the CFD users stay within the Reynolds-Averaged Navier-Stokes (RANS) formalism, deploying the two-equation eddy viscosity models. Owing this wide usage to the numerical stability and robustness, mostly used are the $k-\varepsilon$ model (Jones and Launder, 1972), $k-\omega$ model (Wilcox, 1998), or their modifications. Although in the past many advanced turbulent models have been proposed, ranging from the Reynolds Stress Model (Launder et al., 1975) to the $\overline{v}^2-f$ elliptic relaxation eddy viscosity model (Durbin, 1991), for complex engineering problems they are seldom used because of the concerns about their numerical stability and computational demands.

The turbulence model presented in this paper is based on Durbin’s $\overline{v}^2-f$ approach, as it improves the accuracy of results. Unlike the standard $k-\varepsilon$ model, which due to its isotropic nature tends to over-predict the turbulent mixing (deteriorating thus the accuracy of simulation predictions), the $\overline{v}^2-f$ model accounts for the near-wall Reynolds stress anisotropy and kinematic wall blocking effect. However, while $\overline{v}^2-f$ model accurately captures the most important wall related flow characteristics (separation, reattachment, impingement, streamline curvature), it proved to cause numerical instabilities, especially in the calculations on meshes of poor quality (which are typical for complex wall-bounded building flow problems).

Aiming at improving the robustness and stability of Durbin’s original model, while retaining its quality of results, the $\zeta-f$ variant was derived by Hanjalić and Popovac (2004) by introducing the dimensionless turbulent velocity scale $\zeta=\overline{v}^2/k$ (ratio between $\overline{v}^2$ the fluctuating velocity component normal to the streamlines, and $k$ the turbulent kinetic energy) in the definition of the dynamic eddy viscosity $\mu$:

$$\mu=\rho C_\mu k \zeta T$$

where: $C_\mu=0.22$ is the turbulent viscosity constant, $\rho$ is the fluid density, and $T$ is the turbulent time scale (Eq.(7), requires special near-wall treatment).

In the original $\zeta-f$ model the elliptic relaxation function has the non-zero wall boundary condition, and this can cause numerical problems when considering the implementation of the model into a general purpose RANS-based fluid flow solver. This paper presents the modification of the original $\zeta-f$ model (to obtain zero wall boundary condition for turbulent quantities), as well as its implementation and application in the analysis of realistic external building flows cases.
TURBULENCE MODEL

The elliptic relaxation eddy viscosity turbulence modeling approach accounts for the anisotropy of the Reynolds stress tensor in the wall-normal direction by considering the fluctuating velocity component in the wall-normal direction. Durbin used \( \nu \) as the characteristic velocity scale in the definition of the dynamic turbulent viscosity (Eq.1), since \( \nu \) defines the dominant scale in the near-wall region. This is contrary to the standard \( k-\epsilon \) model, where the characteristic velocity scale is calculated from \( k \) which is isotropic scalar. However, there is a price to be paid for this improvement: compared to the standard \( k-\epsilon \) model, Durbin's approach requires two extra equations to be solved - in addition to the common ones for \( k \) the turbulent kinetic energy and \( \epsilon \) the turbulent kinetic energy dissipation rate:

\[
\frac{\partial \rho k}{\partial t} + \frac{\partial \rho \mathbf{v} \cdot \mathbf{u}}{\partial x_i} = P_k - \rho \epsilon - \frac{\partial}{\partial x_j} \left[ \frac{\mu_s}{\sigma_v} \frac{\partial \mathbf{v}}{\partial x_j} \right] + \frac{\partial}{\partial x_j} \left[ \frac{\mu_s}{\sigma_v} \frac{\partial \epsilon}{\partial x_j} \right]
\]

where \( P_k = \mu_s S^2 \) is the production of the turbulent kinetic energy, \( S = \sqrt{2S_{ij}S_{ij}} \) is the modulus of the mean rate-of-strain tensor, \( \sigma_v = 1.0 \) and \( \epsilon = 1.3 \) are the standard wall boundary condition for the dissipation rate is expressed in terms of \( k \) and the wall distance (denoted as 'y*') which are evaluated in the near-wall cell center (denoted with the subscript 'P'):

\[
\epsilon = 2 \nu k / y^*.
\]

Durbin derived the transport equation for \( \nu \) from the equation for the Reynolds stress component \( \nu \mathbf{v} \) in the direction normal to the wall: inserting \( i = j = 2 \) yields \( \nu \mathbf{v} = \nu \mathbf{v} \) which is in general case assumed to be normal to the streamlines rather than normal to the wall boundaries. Lien and Kalitiz (2001) proposed the expanded \( \nu \mathbf{v} \) transport equation:

\[
\frac{\partial \rho \nu \mathbf{v}}{\partial t} + \frac{\partial \rho \mathbf{v} \cdot \mathbf{u}}{\partial x_i} = \rho f - \rho \mathbf{v} \cdot \frac{\partial \mathbf{v}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \frac{\mu_s}{\sigma_v} \frac{\partial \nu \mathbf{v}}{\partial x_j} \right]
\]

in which all terms describing the interaction between velocity field and pressure field, which are unclosed in the original transport equation for the \( \nu \mathbf{v} \) Reynolds stress component, are modeled through the \( f \) relaxation function, and \( N_\gamma \) is the model constant which is chosen such that the zero boundary condition \( f = 0 \) is obtained. Namely, the definition of \( \nu \mathbf{v} \) and the no-slip requirement yield \( \nu \mathbf{v} = 0 \) at the wall. However, from Eq.(3) the expression for the wall boundary condition \( f_w \) is derived, which is non-zero in general case.

Further to Eq.(2) for the \( k \) and \( \epsilon \) set of equations, the first additional transport equation which needs to be solved in the present modeling framework is the one for the normalized velocity scale \( \zeta \). The derivation of this equation follows from its definition: by combining the transport equations for \( k \) (Eq.2) and \( \nu \) (Eq.3), the mathematical manipulation yields:

\[
\frac{\partial \rho \zeta}{\partial t} + \frac{\partial \rho \mathbf{v} \cdot \mathbf{u}}{\partial x_i} = \rho f - \frac{\mathbf{v} \cdot \frac{\partial \mathbf{v}}{\partial x_i}}{k} + \frac{\partial}{\partial x_j} \left[ \frac{\mu_s}{\sigma_v} \frac{\partial \zeta}{\partial x_j} \right]
\]

where \( \sigma_v = 1.2 \) is the turbulent Prandtl number, and \( N_\epsilon \) is the model constant which is to be defined from the zero boundary condition for the relaxation function. The \( \zeta \) definition and the no-slip wall requirement yield \( \zeta_w = 0 \) the zero value at the wall.

In the \( \zeta \) equation (Eq.4) the relaxation function \( f \) models the pressure-rate-of-strain effects, by solving the elliptic equation of Helmholtz type. The rationale for this step Durbin found by noting that the kernel of the Poisson equation of the fluctuating pressure is actually a Green's function for the modified Helmholtz equation. Hence, the solution of the elliptic equation, with the appropriate pressure-rate-of-strain model in the source, is the model for the pressure-rate-of-strain effects. The present work uses the combination of models proposed by Speziale et al. (1991) and Rotta (1951), and in the conjunction with Eq.(4) the final form of the elliptic equation for the relaxation function \( f \) reads:

\[
L \frac{\partial f}{\partial x_i} = f - \frac{1}{T} \left[ (C_\mu - 1 + C_\rho \frac{P_k}{\rho \epsilon}) \frac{\partial \zeta}{\partial x_i} + 2 \nu \frac{\partial^2 \zeta}{\partial y^*} \right]
\]

with \( C_\mu = 1.4 \) and \( C_\rho = 0.65 \) the model constants. The above Eq.(5) is the second additional equation to be solved in the present modeling framework. The wall boundary condition for \( f \) is obtained from Eq.(4):

\[
f_w = (N_\gamma - 2) \frac{2 \nu \zeta}{y^*}
\]

By choosing \( N_\gamma = 2 \) the zero wall boundary condition \( f_w = 0 \) is obtained, and this is the value used in the present model modification. With \( N_\gamma = 1 \) the obtained set of equations retrieves the form of the original model.

To complete the set of equations for this modified model, the time scale \( T \) and the length scale \( L \) are defined such that the singularities (which could cause an unrealistic behavior of the model) are prevented. To this
purpose $T$ and $L$ are limited from below by the respective Kolmogorov values, and the realizability constraints limit them from above:

$$T=\max \left[ \min \left( \frac{k}{\epsilon} \cdot \frac{C_f}{\sqrt{6} C_L} \right), C_v \frac{\nu^{1/2}}{\epsilon} \right]$$

$$L=C_L \max \left[ \min \left( \frac{k^{1/2}}{\epsilon} \cdot \frac{C_f}{\sqrt{6} C_L} \right), C_v \frac{\nu^{1/2}}{\epsilon} \right]$$

(7)

where $\nu=\mu/\rho$ is the kinematic molecular viscosity of the fluid, and the model constants are $C_f=0.6$, $C_L=6.0$, $C_v=0.36$ and $C_a=50$.

All coefficients used for the present model modification have the same values as those used in its parent $\zeta-f$ formulation, except the $C_o$, which was fine tuned to accommodate for the modification of the boundary condition. For a clear distinction between the parent model and the present modification, however, the set of equations defined by Eqs.(1), (2), (4) and (5), together with (6), (7) and $N=2$ is referred to as the $\zeta-f_0$ model. The subscript "0" is used to stress the point that in this model the zero wall boundary condition holds for the additional turbulent quantities, and not to denote a new variable. Clearly, the $\zeta-f_0$ and $\zeta-f$ formulations are physically identical, therefore the quality of their predictions is the same. However, due to less restrictive boundary conditions $\zeta-f_0$ model has better numerical robustness and computational stability.

The last paragraphs in this section present the details of the model implementation in case of commercial CFD package FLUENT (2007). The starting point for the implementation of the $\zeta-f_0$ model into RANS-based CFD fluid flow solver is the generic transport equation of the standard form. For an arbitrary passive scalar $\phi$ (e.g. additional turbulent quantities or chemical species) this equation can be written as:

$$\frac{\partial \rho \phi}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho f \phi + \Gamma_\phi \frac{\partial \phi}{\partial x_i} \right) = S_\phi$$

(8)

and by solving this equation in the course of the numerical simulation the additional passive scalars, which in FLUENT are called the User Defined Scalars (UDS), can be introduced into numerical simulation.

The generic transport equation of the standard form (Eq.8) consists of four terms: the local rate of change which is activated for the unsteady flow calculations only, the convection for which the flux function $F_\phi$ is to be set, the diffusion for which the diffusivity coefficient $\Gamma_\phi$ is to be defined, and finally the source term $S_\phi$ on the right hand side of Eq.(8). For two additional UDS needed for the $\zeta-f_0$ model, Eq.(8) has to be recast into the form given by Eq.(4) and Eq. (5) for $\zeta$ and $f$ respectively, and their transport equations have to be solved for all fluid zones of the solution domain. In the case of the $\zeta$ equation (Eq.4), this means that the mass flow rate is selected for the flux function $F_\zeta=\rho U_i$, and the effective viscosity $\Gamma_\zeta=\mu+\mu_\nu$ is specified for the diffusivity coefficient. Since the equation for $f$ (Eq.5) does not contain the local rate of change term nor the convection term, none is selected for the flux function and the unsteady term. Comparing Eq.(8) and Eq.(5) it can be concluded that the turbulent length scale squared $L^2$ is the diffusivity coefficient in the $f$ equation. In the actual implementation of the model, however, the entire $f$ equation is divided by $L^2$ since it stays outside of the Laplacian. Hence the actual diffusivity coefficient is set to unity and $L^2$ is transferred into the source term $S_f$. The $\zeta$ and $f$ source terms, required for the $\zeta-f_0$ model implementation, are summarized in the Table 1.

<table>
<thead>
<tr>
<th>$S_\zeta$</th>
<th>$S_f$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho f - \frac{\zeta}{k} (p_t + \rho \epsilon) \frac{\partial}{\partial t} \left( \frac{\partial \phi}{\partial x_i} \right)$</td>
<td>$\frac{1}{L^2} \left[ \frac{1}{7} \left( C_f - 1 + C_k \frac{p_t}{\rho\epsilon} \right) \left( \frac{\zeta - 2}{3} + \frac{\xi}{3} \right) \right]$</td>
</tr>
</tbody>
</table>

Table 1: source terms for $\zeta$ and $f$

The activation of the $\zeta-f_0$ definition of the turbulent viscosity (Eq.1) is actually the key point in the implementation of this new model, as all the turbulence effects are coming into play through the $\mu_\nu$ in the framework of eddy viscosity turbulence modeling. To conclude the discussion about the $\zeta-f_0$ model implementation, there are two remarks. Firstly, it was already mentioned that in the present implementation the entire $f$ equation (Eq.5) is divided by $L^2$ (Eq.10), and consequently the diffusivity coefficient in the $f$ equation is set to unity. Therefore it is important, from the numerical point of view, to prevent the turbulent length scale $L$ becoming zero (the turbulent time scale $T$ is treated the same). And secondly, further to the initial conditions for $k$ and $\epsilon$ which are well established (e.g. through the turbulent intensity and the viscosity ratio), the initial conditions for $\zeta$ and $f$ are also very simple. Since zero boundary condition at the wall applies for $f$ (Eq.6), it is reasonable also to initialize the $f$ field with zero. As for the normalized velocity scale, in the equilibrium state (when the $\zeta-f_0$ model reduces to the standard $k-\epsilon$ model) $\zeta$ obtains the value $2/3$. Hence $\zeta_{in}=2/3$ is a recommended $\zeta$ initialization.
EXTERNAL BUILDING FLOWS

The derivation of the $\zeta-f_0$ model and its implementation into the commercial CFD package FLUENT has already been reported elsewhere (Popovac, 2010), together with its validation on the several flow test cases. At the beginning of this section, therefore, Fig.1 presents only the selected results for the heat transfer coefficient in two generic test cases: backward-facing step and impinging jet flow. In the rest of the section, however, the $\zeta-f_0$ and $k-\epsilon$ results are compared for the case of the flow in a realistic urban environment. Two aspects of the external building flows are analyzed: the characteristics of the flow at the pedestrian level between buildings in a realistic urban environment, and the flow characteristics in the vicinity of a single building of realistic shape.

![Figure 1: distribution of heat transfer coefficients along the bottom heated wall - Stanton number for backward-facing step (a), and Nusselt number for impinging jet (b), obtained with the $\zeta-f_0$ model and the standard $k-\epsilon$ model.](image)

The characteristics of the flow at the pedestrian level between buildings in realistic urban environment is presented here as the first (urban scale) aspect of the numerical analysis of external building flows using the $\zeta-f_0$ model. The realistic urban area (commercial and residential neighborhood in Vienna, Austria), shown in Fig.2, consists of several groups of residential buildings spread along the underground line and the underground station, several public buildings, as well as the shopping mall complex of futuristic design (blue).

![Figure 2: geometry of a developed urban organism, with the exponential inlet velocity profile (a), the flow path-lines around the buildings (b), and the contour plot of the velocity magnitude at the pedestrian level (c).](image)

From the comparison between results for two generic test cases, in Fig.1 showed for the standard $k-\epsilon$ model and the $\zeta-f_0$ model, some conclusions are immediately evident. On the one hand, as a consequence of the isotropic definition of the turbulent viscosity (used to calculate the Reynolds stresses), the predictions obtained with the $k-\epsilon$ model are somewhat diffuse and “blurred” in the regions of strong shearing (e.g. separation, reattachment). On the other hand, due to improved description of the near-wall flow behavior, the $\zeta-f_0$ model predictions of the flow characteristics agree well with the experiments, even in the zones of very strong pressure gradients and streamline curvature (e.g. stagnation, impingement). As the external building flow example given below will show, the same model behavior can be observed for the external building flow simulations, as the flow effects such as the separation, reattachment, streamline curvature and impingement are often encountered in these type of flows. This is why turbulence modeling issues require special attention for the simulations of external building flows.
For the flow simulations of such a developed urban organism it is important to have full geometrical representation in the numerical domain: exact building shape, columns, and alike. The simulation setup is presented in Fig.2: the wind was simulated by imposing the exponential velocity profile on one of the domain boundary faces, and the overall flow pattern for this case can be identified from the flow path-lines around the buildings. In this paper only the flow simulation for a single wind flow direction is presented. However, for the detailed wind comfort study the flow situation is assessed by performing a series of flow simulations for all relevant wind directions, in which the representative wind profile is imposed for each wind direction. Simulations of this kind are typically performed in order to assess either the wind comfort around the buildings at the pedestrian level, or the wind load for the building statics. The purpose of such simulation is to capture the flow characteristics around given building, with high level of geometrical details. High complexity of the geometry, however, can cause problems in the numerical simulation (especially if advanced turbulence models are used), and thus the tendency in computation is to simplify the geometry as much as possible. On the other hand, in the present case this high level of the geometry complexity is the simulation requirement, and exactly this is where the robustness and the stability of the turbulence model come into play. The presented computations, performed using the $\zeta-f_0$ model, showed no instabilities or numerical difficulties of any kind. The improvements in the accuracy of numerical simulations, obtained through better physical modeling, are only on the cost of 30% increase in the computational time in comparison to the $k-\varepsilon$ model.

![Contour plot of the velocity magnitude at the pedestrian level](image1)

(a)

![Contour plot of the pressure at the building surfaces](image2)

(b)

Figure 3: contour plot of the velocity magnitude at the pedestrian level ($2 \text{ m }$ above the ground), obtained with the $\zeta-f_0$ model (a) and the standard $k-\varepsilon$ model (b).

![Contour plot of the static pressure at the building surfaces](image3)

(a)

![Contour plot of the static pressure at the building surfaces](image4)

(b)

Figure 4: contour plot of the pressure at the building surfaces on the windward side, obtained with the $\zeta-f_0$ model (a) and the standard $k-\varepsilon$ model (b).

The contour plots of the velocity magnitude in the cut-plane at the height of $2 \text{ m }$ above the ground, shown in Fig.3, reveal the principal flow pattern. The high
velocity zones are found at the edges of high buildings, in the passages between them and under the bridges, whereas the low velocity zones are identified at the leeward building side (protected by buildings) where the recirculation flow pattern occurs, or in the covered building parts (building entrance) where the stagnation flow occurs. The contour plots of the surface pressure distribution, presented in Fig.4, are directly related to the obtained velocity field. All windward faces of buildings have increased static pressure, with the highest static pressure exerted on the most opened building surfaces (the flow stagnation). The inner corners and the leeward faces of the buildings are well protected, and under-pressure fields occurs (the flow recirculation). The obtained results indicate that no problems are expected regarding the wind comfort.

Although the flow simulations of such a large scale are hard to quantify the conclusions, some trends in the model predictions can be identified. The overall flow pattern obtained with the $\zeta-f_0$ and $k-\epsilon$ model, as well as the minimum and maximum values are predicted approximately the same, although close comparison between the results for the velocity magnitude (Fig.3) and the pressure distribution (Fig.4) will show the differences in details around edges. More pronounced is the difference in predicting turbulent quantities. As seen in Fig.5, the turbulent kinetic energy predicted by the $k-\epsilon$ model is significantly bigger than that obtained with the $\zeta-f_0$ model, which also explains the difference in the velocity field (the over-prediction of $k$ is well known deficiency of the standard $k-\epsilon$ model). Even if the turbulent mixing is not the focal point of the wind comfort analysis, it is important e.g. for the simulations of the pollution dispersion. This shows the importance of the turbulence modeling for the external building flow simulations.
The flow characteristics in the vicinity of a single building of realistic shape is presented here as the second (building scale) aspect of the external building flow numerical analysis using the \( \zeta-f_0 \) model. In this case the accent is on better understanding of the flow conditions in the immediate vicinity of the building, which can be important e.g. for defining certain wind protection measures for the locations on buildings which are critical from the wind comfort point of view (e.g. terraces or building entrance). Another application, presented here in more details, is the analysis of the optimal positioning of the Small Wind Turbines (SWT) integrated in buildings. For this purpose the required level of information details regarding the flow characteristics goes much beyond the velocity and pressure field distribution, because the turbine operation depends also on the turbulence intensity and the flow shear stress. Hence, for a detailed analysis of external building flows, an appropriate turbulence model has to be used.

Unlike the flow simulations on a large (urban) scale, where the velocity, pressure or concentration are usually the flow quantities of practical interest, for the small (building) scale simulations also the turbulent quantities are of great importance because they are determining the local differences in the exact flow pattern. Furthermore, the power output of SWT is defined not only by the wind velocity, but also the turbulence intensity level. This is due to the shear effects originating from the strong turbulent interactions (generally related to high rate of strain) that are taking place in highly turbulent flow. Therefore appropriate turbulence model is not only computationally stable, but also accurate in predicting turbulent quantities in complex flows.

The analysis of the exact positioning of SWT integrated in buildings is performed by inspecting the flow characteristics at the desired location, in order to see what are the wind characteristics at the location where SWT would be mounted. The contour plots in Fig.6 show the velocity magnitude in the cut-surface equidistant from the building roof (4 m), in order to identify the zones of high wind velocity. In the wakes of the roof protrusions (e.g. chimneys, equipment rooms) the velocity is reduced, while the building edges are the zones of flow acceleration due to the deflection of the flow by the building. Note, however, the difference in the velocity predictions between the \( \zeta-f_0 \) and \( k-\epsilon \) model. The high velocity zone across the roof diagonal is much more diffuse with the \( k-\epsilon \) model, contrary to "sharper" \( \zeta-f_0 \) predictions. Accordingly the maximum value predicted with the \( \zeta-f_0 \) model is higher than the one with the \( k-\epsilon \) model.

In case of the turbulent kinetic energy, the contour plots given in Fig.7 show that the difference between the \( \zeta-f_0 \) and \( k-\epsilon \) model predictions is not only the distribution but much more the obtained values. It has been already mentioned that the \( k-\epsilon \) model can produce too high \( k \) levels, and this is evident in the flow region above the building roof. On the other hand, however, the \( \zeta-f_0 \) distribution of \( k \) in the cut-surface equidistant from the building roof (4 m) follows closely the shear regions around the building edges and behind the roof protrusions. With the \( k-\epsilon \) model the source of the production of \( k \) is the same (the flow shear), but immanent to the model is to produce unrealistically high \( k \) values.

Very important for the operation of SWT is the rate of strain, shown in Fig.8. Together with the turbulent kinetic energy, it determines the aerodynamic resistance of SWT, hence it directly influences the energy performance of SWT. The distribution of the rate of strain, shown in Fig.6 in the cut-surface equidistant...
from the building roof (4 m), indicates that the steepest velocity gradients are found around the building edges. Although the velocity magnitude is also increased in that region, the efficiency of SWT is reduced there due to unfavorable turbulent flow characteristics. Both the $\zeta-f_o$ and $k-\epsilon$ model predict the same location for the highest rate of strain, but $k-\epsilon$ model gives wider zone of increased rate of strain, compared to the $\zeta-f_o$ model. This was expected from the turbulence model that performs over-diffusively.

**CONCLUSION**

The aim achieved by the derivation of the $\zeta-f_o$ model is the definition of the advanced and robust eddy viscosity turbulence model with elliptic relaxation. This model is more accurate than the most widely used standard $k-\epsilon$ model, as it accounts for the Reynolds stress near-wall anisotropy. The easy implementation of the $\zeta-f_o$ model into RANS-based CFD code is simple and straightforward, so that this “upgrade” of standard $k-\epsilon$ model is fully justified for the simulations of complex wall-bounded building flows. With the zero wall boundary condition for the additional turbulent quantities, and the identical form of the additional equations to be solved, there is a minimal programming effort required in order to have all the advantages of the improvements that the turbulence modeling concept with elliptic relaxation brings in the numerical analysis of external building flows.

The implementation of the $\zeta-f_o$ model is described here on the example of the commercial CFD package FLUENT, where the standard $k-\epsilon$ model is already present. This implementation requires the use of two additional User Defined Scalars, for which the transport equations are solved. In both equations the diffusivity coefficient has to be specified, the convection term and the local rate of change have to be appropriately chosen, and zero constant value boundary conditions specified at all solid boundaries. Finally, the source terms have to be defined through User Defined Functions, and the turbulent viscosity is appropriately modified in order to take into account the near-wall turbulent effects. The only drawback of this “upgrade” to the $\zeta-f_o$ model is approximately 30% increase in the computation time, which is much less then the increase one would expect (100% more compared to the standard $k-\epsilon$ model, since two additional equations are solved). Hence the increased computational demand is worth the increased quality of results.

The flow predictions and the computational performance of the $\zeta-f_o$ model was tested on the external building flow case, with the focus both on large (urban) and small (building) scale. As an example of the complex wall-bounded flow in a realistic urban environment, the flow in an entire neighborhood was calculated with the $\zeta-f_o$ model. Also the flow characteristics around a single building of realistic shape were investigated in more detail. The obtained results using the $\zeta-f_o$ model show good quality for predicting the flow characteristics, especially concerning turbulent quantities (the turbulent kinetic energy). Furthermore, the computational stability and robustness of the $\zeta-f_o$ model was very good.
ACKNOWLEDGMENT

The financial support from Österreichische Forschungsförderungsgesellschaft mbH (FFG) for the presented work is gratefully acknowledged (project “IPPONG“, project number 821910).

REFERENCES


