THEORETICAL AND APPLIED MECHANICS
Volume 45 (2018) Issue 2, 139–149 DOI: https://doi.org/10.2298/TAM171201009N

ON THE INFLUENCE OF TURBULENT KINETIC ENERGY LEVEL ON ACCURACY OF $k - \varepsilon$ AND LRR TURBULENCE MODELS

Djordje M. Novković, Jela M. Burazer, Aleksandar S. Ćoćić, and Milan R. Lečić

Abstract. This paper presents research regarding the influence of turbulent kinetic energy (TKE) level on accuracy of Reynolds averaged Navier–Stokes (RANS) based turbulence models. A theoretical analysis of influence TKE level on accuracy of the RANS turbulence models has been performed according to the Boussinesq hypothesis definition. After that, this theoretical analysis has been investigated by comparison of numerically and experimentally obtained results on the test case of a steady-state incompressible swirl-free flow in a straight conical diffuser named Azad diffuser. Numerical calculations have been performed using the OpenFOAM CFD software and first and second-order closure turbulence models. TKE level, velocity profiles and Reynolds stresses have been calculated downstream in four different cross sections of the diffuser. Certain conclusions about modeling turbulent flows by $k - \varepsilon$ and LRR turbulence models have been made by comparing the velocity profiles, TKE distribution and Reynolds stresses on the selected cross sections.

1. Introduction

First and second-order closure turbulence models have significant usage in numerical calculations of turbulent flows. The accuracy of these models depends significantly on Reynolds number values and type of analysed turbulent flow. It is impossible to reach good accuracy by the use of these models on a wide range of Reynolds numbers and wide classes of turbulent flows. The local Reynolds number values and type of local turbulent flow are strict variably in many cases of turbulent flow in engineering and nature. For example, in the flow in a straight conical diffuser there is wide-scale change of the local Reynolds number values. Generally, the flow in a straight conical diffuser consists of swirl and swirl-free local zones, shear and shear-free local zones, low and high values of local Reynolds number zones etc. Further, flow in a straight conical diffuser has strong downstream adverse pressure

2010 *Mathematics Subject Classification:* 76F10.

Key words and phrases: CFD, diffuser, OpenFOAM, RANS, turbulent kinetic energy.

The paper is presented at the 6$^{\text{th}}$ Serbian Congress in Theoretical and Applied Mechanics (19—21 June 2017), Minisimposia Turbulence. The paper is a part of the collection of papers from the Congress published in the issue 1 of the same volume.

gradient (APG) generated by the shape of diffuser. For these reasons, flow in a straight conical diffuser is a good test case for turbulence models. It is challenging to find the main influences of the decrease in accuracy of used turbulence model in certain flow zones of straight conical diffuser. Okwuobi and Azad [1, 2] analyzed the flow in a straight conical diffuser experimentally. The Azad's diffuser has a spreading angle of 8°. Okwuobi and Azad analyzed the swirl-free flow conditions in this diffuser under low intensity of boundary layer separation (BLS). The experimental results from this research have wide usage as a validation data base for numerical computations.

The swirl-free flow in Azad's diffuser can be numerically calculated by the use first-order closure model like standard $k-\varepsilon$ turbulence model. The first-order closure models leads to failure in flows where there is significant anisotropy of the normal stresses. They can not account for streamline curvature effects and can't model secondary and swirling flows [3]. The fundamental concepts of the numerical flow simulations in the straight conical diffusers using $k-\varepsilon$ turbulence model are given in USAF Research report AEDC-TR-76-15 [4]. In the case of swirl-free flow in a straight conical diffuser there is no significant streamline curvature. Further, in the central coaxial zone about the axis of diffuser there is no significant anisotropy of the Reynolds stresses, because influences of wall effects are minor in this zone. For these reasons it can be expected that $k-\varepsilon$ turbulence model gives good results here. Yongsen et al. [5] performed 2D numerical flow simulations in the Azad's diffuser using standard $k-\varepsilon$ turbulence model and coarse computational mesh. They achieved good results under assumption of axisymmetric steady-state flow conditions in this diffuser. Zhu and Shih [6] performed numerical flow simulations under the same conditions as those done by Yongsen et al. but they used the anisotropic $k-\varepsilon$ turbulence model. They obtained slightly better results in velocity profiles predictions near the diffuser axis compared to results obtained by isotropic $k-\varepsilon$ model. Certain disadvantages in usage of the $k-\varepsilon$ turbulence model for numerical prediction of flows in diffusers were analyzed by Armfield and Fletcher [7]. They compared this first-order closure model with two algebraic Reynolds stress models.

The second-order closure models like algebraic Reynolds stress models and full Reynolds stress models partially overcome assumption of isotropic turbulent viscosity, which is serious disadvantage of the first-order closure models based on Boussinesq hypothesis. Full Reynolds stress models (RSM) like Launder-Reece-Rody (LRR) have significant usage in diffuser flow calculations. Karvinen and Ahlstedt [8] performed numerical calculations of the flow in asymmetric diffuser using LRR turbulence model and OpenFOAM CFD codes. They concluded that the LRR turbulence model implemented in OpenFOAM CFD codes gives poor results in this class of flow. In some cases of flow conditions with aproximatelly shear-free flow conditions, the LRR turbulence model might have poor numerical stability caused by instabilities of low Reynolds stress values. Maduta [9] performed calculation of flow in asymmetric diffuser using eddy-resolving full RSM. This model detects instabilities of Reynolds stresses and captures better the influences of the instantaneous character of the turbulent flow. Maduta showed that this model gives better result in velocity profiles and Reynolds stress prediction.

The anisotropy of Reynolds stresses has significant influence on the accuracy of first-order closure models. Radenkovic et al. [10] analysed main types of Reynolds stresses anisotropy in case of swirl flow in a straight pipe. They performed analyses of the Reynolds stresses using invariant theory. Basically, the rate of Reynolds stresses anisotropy depends on values of stresses in isotropic and anisotropic part of Reynolds stress tensor (RST). If isotropic part of the RST has significantly higher values of stresses than anisotropic part, then RST have significantly higer rate of isotropy. This is crucial fact which helps to understand the importance of influence isotropic part of RST on turbulence modeling. Naimely, errors in modelling of anisotropic part of the RST can have significant influences on accuracy of RANS based turbulence models. If components of anisotropic part of the RST have significantly lower values than components of isotropic parts of RST, than errors in modelling of anisotropic part of RST have significantly lower influence on accuracy of RANS based turbulence models. The primary aim of this paper is detection of the influence of components of isotropic part RST on the accuracy of first and second-order closure model. This has been performed on test case of swirl-free flow in Azad's diffuser. Since eddy-resolving full RSM is not implemented in OpenFOAM code, numerical calculation in this paper has been performed using LRR full RSM model.

2. Mathematical models

2.1. Governing equations and turbulence models. The flow that is considered in this paper is incompressible steady-state turbulent flow of a Newtonian fluid. The equations governing this kind of flow are the averaged continuity equation

$$\nabla \cdot \underline{U} = 0,$$

and the averaged Navier–Stokes equation, also known as the Reynolds equations

$$\nabla \cdot (\underline{U} \otimes \underline{U}) = -\nabla(P/\rho) + \nu \cdot \Delta \underline{U} - \nabla \cdot \underline{\underline{R}} \tag{2.1}$$

The dyad $\underline{\underline{R}} = \langle \underline{u} \otimes \underline{u} \rangle$ is the RST. It represents a new unknown in the system of equations which needs modeling in order to calculate turbulent flow. The first-order closure model, named $k-\varepsilon$ and second-order closure model, named LRR have been used for computation analysis presented in this paper.

2.1.1. *The $k - \varepsilon$ turbulence model.* This is two-equation turbulent model that calculates eddy viscosity by eqaution $\nu_t = C_\nu k^2/\varepsilon$, after solving the transport equation for turbulent kinetic energy k, and the transport equation for dissipation ε [11]. Further, here unknown components of RST are then computed by Boussinesq hypothesis equation

$$\underline{\underline{R}} = \tfrac{2}{3} k \underline{\underline{I}} - 2\nu_t \underline{\underline{S}} \tag{2.2}$$

which decompose RST on isotropic and anisotropic part and gives the possibility to close the system of equations (2.1). It is important to say that the anisotropy of RST here comes only from anisotropy of the strain rate tensor, while turbulent viscosity is asummed to be isotropic. Here the turbulent viscosity ν_t can be written as second-order isotropic tensor whose all diagonal components are equal to scalar values of turbulent viscosity ν_t, and all off-diagonal componets are equal zero. In reality, the turbulent viscosity is not fluid characteristic, it depends on velocity field and it is anisotropic because velocity field is anisotropic.

2.1.2. *The LRR turbulence model.* This model does not use Boussinesq hypothesis (2.2) and assumption of isotropic turbulent viscosity. In LRR turbulence model each commponent of RST has own transport equation which is obtained from tensor equation

$$\nabla \cdot [\underline{U} \otimes \underline{\underline{R}}] = \underline{\underline{P}} - \underline{\underline{\varepsilon}} + \underline{\underline{\Pi}} + \underline{\underline{D}} + \nu \Delta \underline{\underline{R}}$$

where terms: $\underline{\underline{P}}$, $\underline{\underline{\varepsilon}}$, $\underline{\underline{\Pi}}$ and $\underline{\underline{D}}$ are calculated as it is described in [12]. These 6 partial differential equations for Reynolds stresses and one partial differential equation for ε forms system of 7 partial differential equations for RST components calculation. Solving of this system requires sinificantly more computation effort comparing to two-equation $k - \varepsilon$ turbulence model. It has good potential to capture anisotropy of RST, but in some cases it suffers from numerical instability. More details about LRR turbulence model can be found in Launder et al. [13].

2.2. Influence TKE on RST components modeling errors. As it can bee seen from equation (2.2) the TKE forms isotropic part of RST and hence it has significant influence on the RST modeling. Generally, the main problem in reliability of RANS based turbulence models is the accuracy of RST components modeling. It is dificult to discover the real power of RANS based turbulence models becaue it is impossible to model RST components without the presence of modeling errors. Further, it is difficult to discover amount of the modeling errors influence by analysis of the final numerical solution, because final numerical solution contain more type of errors such as: RST components modeling errors, errors of flow domain spatial discretisation, errors of convergence of numerical solution etc. However, the RST components modeling errors have the most significant influence on final solution in the most cases of complex turbulent flows. Generally, in tensor index notation, exact value of the RST component can be written as

$$\langle u_i u_j \rangle_{\mathrm{exact}} = \tfrac{2}{3}(k)_{\mathrm{exact}}\delta_{ij} + (b_{ij})_{\mathrm{exact}},$$

where $(b_{ij})_{\mathrm{exact}}$ represents exact value of anisotropic part of RST component. The modelled component of the RST can be written as

$$\langle u_i u_j \rangle_{\mathrm{mod}} = \tfrac{2}{3}(k)_{\mathrm{mod}}\delta_{ij} + (b_{ij})_{\mathrm{mod}}.$$

The difference between exact and modeled value of TKE represents the total error of RST component modeling. Under assumption that the TKE is modeled exactly ($(k)_{\mathrm{mod}} = (k)_{\mathrm{exact}} = k$), the total error of RST component modeling is reduced to difference of anisotropic parts of RST component

$$E = \langle u_i u_j \rangle_{\mathrm{exact}} - \langle u_i u_j \rangle_{\mathrm{mod}} = (b_{ij})_{\mathrm{exact}} - (b_{ij})_{\mathrm{mod}}.$$

From the computational point of view, it is important to estimate the relative error of RST component modelling

$$e = \frac{E}{\langle u_i u_j \rangle_{\mathrm{exact}}} = \frac{(b_{ij})_{\mathrm{exact}} - (b_{ij})_{\mathrm{mod}}}{\frac{2}{3}k\delta_{ij} + (b_{ij})_{\mathrm{exact}}}.$$

From the last equation it can bee seen that the relative error of RST component modeling depends on TKE level. In parts of fluid flow domain with low values of TKE the relative error of RST component modeling has high value

$$e = \lim_{k\to 0} \frac{(b_{ij})_{\mathrm{exact}} - (b_{ij})_{\mathrm{mod}}}{\frac{2}{3}k\delta_{ij} + (b_{ij})_{\mathrm{exact}}} = 1 - \frac{(b_{ij})_{\mathrm{mod}}}{(b_{ij})_{exact}}.$$

In parts of fluid flow domain with high values of TKE the relative error of RST component modeling has very low value

$$e = \lim_{k\to \infty} \frac{(b_{ij})_{\mathrm{exact}} - (b_{ij})_{\mathrm{mod}}}{\frac{2}{3}k\delta_{ij} + (b_{ij})_{\mathrm{exact}}} = 0.$$

According to above theoretical analysis it can be concluded that level of TKE has significant influence on the accuracy of RST components modeling. Performed analysis shows that it is easier to model turbulent flow with higher values of TKE. In parts of fluid flow domain with low values of TKE the usage of second-order turbulence models can results with significant error. This is the consequence of significant influence of big relative errors during RST components modeling process. This can be proved practically on the test case of swirl-free turbulent flow in Azad's diffuser.

3. Geometry and boundary conditions of test case

The geometry of analyzed diffuser is shown in Figure 1. The diffuser has the spreading angle of 8° and short cylindrical parts at the inlet and outlet sections. These cylindrical parts were used in DNS model [14]. They are helpful in the implementation of boundary conditions. The inlet cylindrical part has a diameter $R_{\mathrm{ref}} = 50.8\,\mathrm{mm}$.

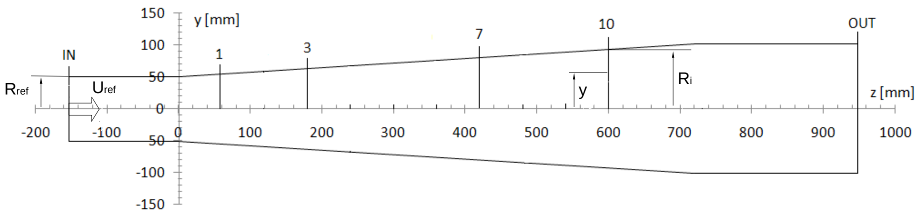


FIGURE 1. Geometry of the diffuser.

The results of numerical simulations in this paper have been compared with the experimental data (cross sections 1, 3, 7, 10) of Okwuobi [1]. The steady-state incompressible swirl-free air flow at Reynolds number Re = 152000 on the diffuser inlet section has been analyzed.

The numerical flow simulations have been performed on a 3D mesh using an open-source software OpenFOAM. The block structured mesh presented in Figure 2 consists of 256224 cells. It has been created using ANSYS ICEM CFD software. The mesh has been imported in OpenFOAM software using application cfx4ToFoam. The experimental values of the velocity field (fully developed turbulent pipe flow) have been used as a boundary condition on the INLET. Fixed values of the TKE and the energy dissipation rate taken from [5] have been also set on the INLET. On the OUTLET, the kinematic pressure has been set to $0\,\mathrm{m^2 s^{-2}}$, while for the other quantities a zeroGradient boundary condition has been used. A no-slip boundary condition for the velocity and the zeroGradient boundary condition for the kinematic pressure have been set on the diffuser's wall. In the wall region, where higher gradients of a certain physical quantity are expected, a mesh grading technique has been introduced.

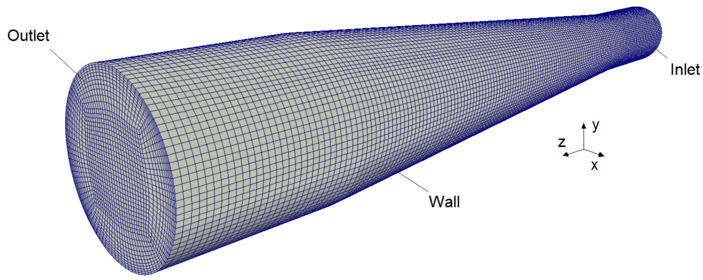


Figure 2. Computational mesh of the diffuser.

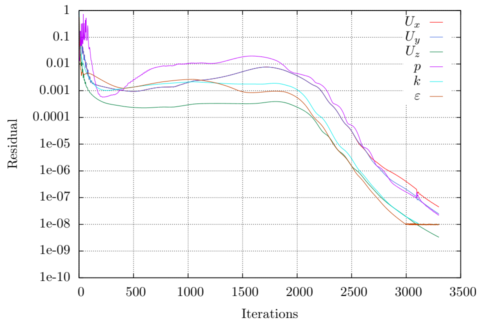
The first layer of thickness of boundary layer mesh has been carefully set up. The values of y^+ are 30 to 60 in almost all boundary layer of the computational domain have been set. The Gauss linear scheme has been used for discretization of the convection terms. Convection terms in turbulence equations have been discretized by the upwind scheme. Since these are steady-state computations, under-relaxation procedure has been used in order to improve the stability of the calculations. The air is a working medium with kinematic viscosity of $\nu = 1.545 \times 10^{-5}\,\mathrm{m^2/s^2}$.

4. Convergence of the solution

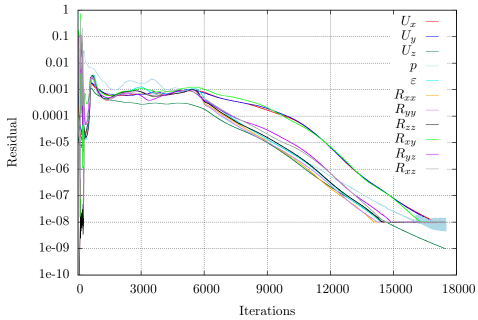
All calculations have been performed using the simpleFoam solver which is based on the SIMPLE algorithm to solve the discretized equations. In some cases of the SIMPLE algorithm usage high number of iterations is needed in order to reach low values of final residuals. By comparing convergence on Figures 3a and 3b it can be seen that LLR turbulence model requires significantly greater number of iterations compared to $k - \varepsilon$ model in order to reach the same values of final residuals. Both of the used turbulence models in this paper have had stable convergence and they have reached low values of final residuals. It is visible that there have been certain instabilities in the residual values at the end of the convergence process, but this is not so significant.

5. Results and discussion

Nondimensional velocity profiles obtained by numerical calculations have been compared with experimental data [1] in Figure 4. As it can be seen from this figure there are significant differences between velocity profiles obtained with used turbulence models, especially in the coaxial part of the flow, around diffuser's axis.



(A)



(B)

FIGURE 3. Convergence of: (A) $k-\varepsilon$ model and (B) LRR model.

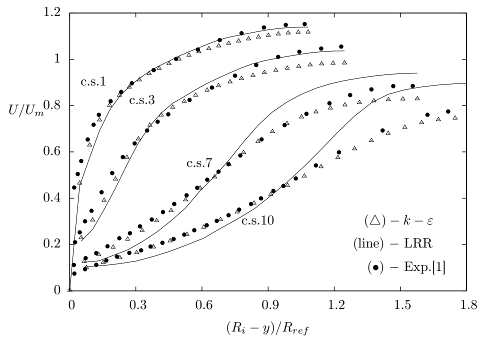


FIGURE 4. Radial distribution of dimensionless axial velocity.

The LRR turbulence model gives better results than the standard $k-\varepsilon$ model only in the first two cross sections (1, 3), where the flow is strongly influenced by the fully developed turbulent flow at diffuser inlet section. The results obtained by $k-\varepsilon$ and LRR turbulence model have excellent agreement with experimental data only in the central flow zone of the cross sections 1 and 3.

Downstream, in cross sections 7 and 10, where significant disturbances caused by flow deceleration are present, standard $k-\varepsilon$ model gives better results than LRR turbulence models. The best agreement between numerical and experimental results across the whole flow domain is obtained by standard $k-\varepsilon$ turbulence model. The standard $k-\varepsilon$ turbulence model has lower sensitivity on disturbances which amplify the turbulent flow unsteadiness. One of the most significant parameters in turbulence modeling is TKE which describes the isotropic part of the Reynolds stress tensor as it has been shown above.

From Figures 4 and 5 it can be seen that all of the used turbulence models give good results only in the flow zones where TKE has maximum values. In these zones the isotropic part of the Reynolds stress tensor has a more significant influence on turbulence modeling, compared to the anisotropic part. In the flow zones where

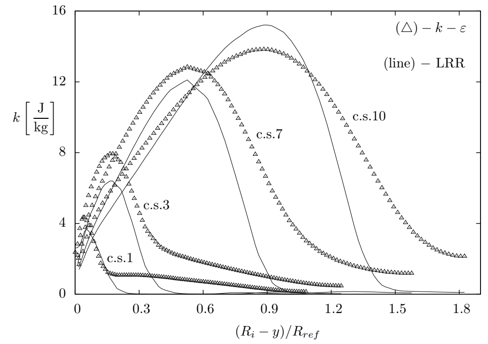
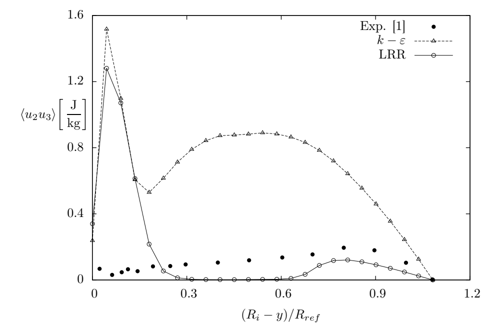


FIGURE 5. Distributions of the turbulent kinetic energy in the radial direction.



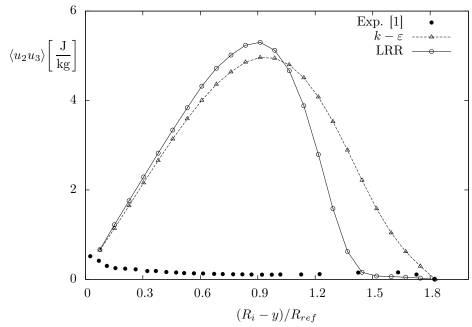


FIGURE 6. Turbulent stress distributions in: (left) cross section 1 and (right) cross section 10.

TKE has minimum values isotropic and anisotropic parts of the Reynolds stress tensor have similar contribution to turbulence modeling.

In the coaxial flow zone of the analyzed diffuser, especially in the cross sections 7 and 10, the components of anisotropic part of the Reynolds stress tensor have low and unstable values. The TKE also has low values in these parts of the flow. Since there is no compensation of these low and unstable values of stresses by the isotropic part of Reynolds stress tensor, these unstable values of stresses have significant influence on the turbulence modeling. In LRR turbulence model these low and unstable values of stresses are unreliably modeled due to the presence of instability of variable in each of six stress transport equations. In the first two considered cross sections (1, 3) in the coaxial part of the flow, components of anisotropic part of Reynolds stress tensor also have low values, but there is higher stability of stresses here under the strong influence of fully developed turbulent flow at diffuser entry. Hence LRR model gives good results here.

In the class of the flow considered in the paper, Reynolds stress $\langle u_2 u_3 \rangle$ has the highest value, which has a significant influence on the flow modeling. According to

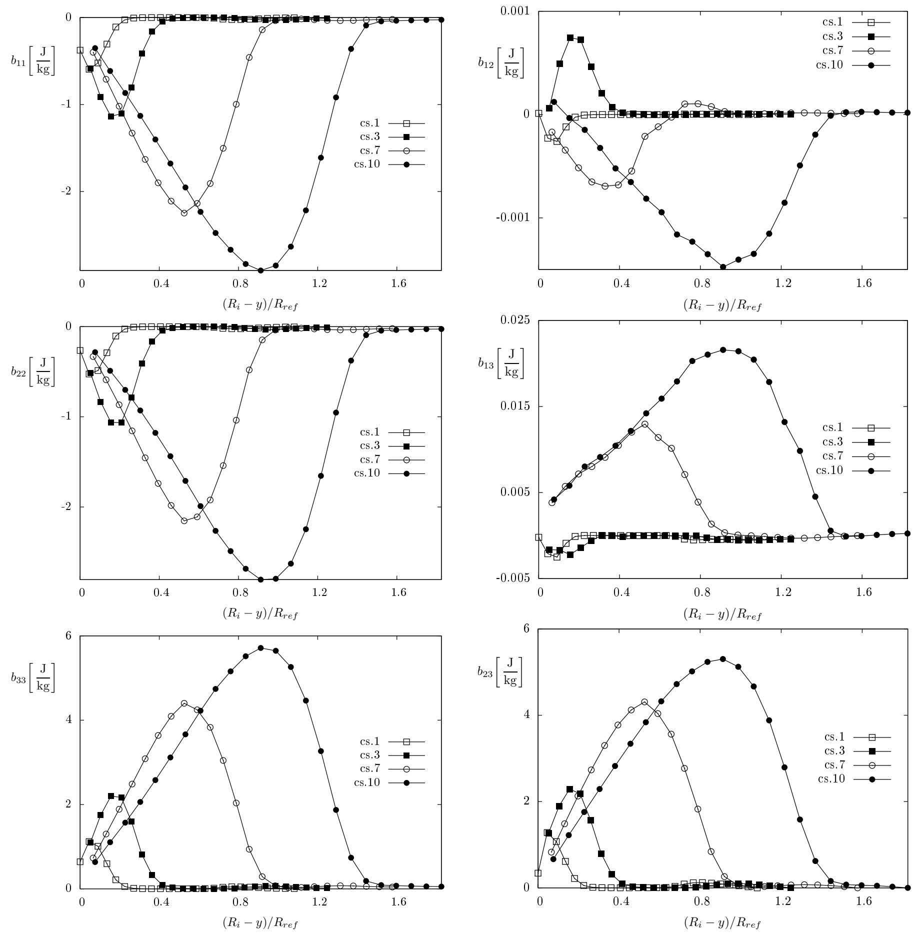


FIGURE 7. Distributions components b_{ij} of anisotropic part of RST calculated by LRR full RSM.

Figure 6 this stress has zero values on the axis of the diffuser. Other stresses and TKE have low values here as well. Hence, Reynolds stress tensor is modeled with low and unstable values of turbulent stresses here. These low and unstable values of turbulent stresses cause the amplification of the modeling errors in this flow zone. At the same time, Figure 7 shows that all components of anisotropic part of RST also have low values in the coxial part of the flow domain. Since stresses have low values in this area of the flow, stress modeling has significant impact on modeling accuracy. This leads to big relative errors in stress modeling. It is interesting that the velocity profiles have the best agreement with the experimental data in zones where dominant off-diagonal Reynolds stress has the biggest discrepancy with the experimental data. The reason for this kind of behavior is that TKE has maximum value here and it compensates the influence of off-diagonal stresses discrepancy.

6. Conclusions

According to the obtained results it can be concluded that both of the used models in this paper have high sensitivity in flow zones of approximately shear-free flow conditions with low values of TKE. In such flow zones the RST is modeled with low and unstable values which can cause significant modeling errors. The TKE has significant stabilization effect on the errors of Reynolds stress tensor modeling and it is easier to model turbulent flows that have higher values of TKE. This paper also shows the advantage of the $k-\varepsilon$ turbulence model usage in flows with shear-free flow conditions. Standard $k-\varepsilon$ turbulence model has lower sensitivity on instability of Reynolds stresses, since it has a simpler method for Reynolds stress modeling. In order to increase the accuracy of LRR full RSM it will be interesting to perform a detailed research regarding the TKE low value influence on the accuracy of the Reynolds stresses tensor modeling.

Acknowledgments. This work was funded by the grant from the Ministry of Education, Science and Technological Development, Republic of Serbia through project TR33046 and TR35046, what is gratefully acknowledged.

References

1. P. A. C. Okwuobi, *Turbulence in a Conical Diffuser With Fully Developed Flow at Entry*, Ph.D. thesis, Department of Mechanical Engineering, Winipeg, Manitoba, October 1972.
2. P. A. C. Okwuobi, R. S. Azad, *Turbulence in a conical diffuser with fully developed flow at entry*, J. Fluid Mech. **57** (1973), 603–622.
3. M. Woelke, *Eddy viscosity turbulence models employed by computational fluid dynamic*, Transaction of the Institute of Aviation **191** (2007), 92–113.
4. J. C. Chien, *Numerical Analysis of Turbulent Separated Subsonic Diffuser Flow*, USAF Research report AEDC-TR-76-15, Arnold Air Force Station, Tennessee, 1977.
5. H. Yongsen, T. Kobayashi, Y. Morinishi, *Numerical prediction of turbulent flow in a conical diffuser using $k-\varepsilon$ model*, Acta Mech. Sin. **8** (1992), 117–126.
6. J. Zhu, T. H. Shih, *Calculations of Diffuser Flows With an Anisotropic model*, NASA Contractor Report 198418, Cleveland, Ohayo, November 1995.
7. S. W. Armfield, C. A. J. Fletcher, *Comparison of and algebraic Reynolds stress models for swirling diffuser flow*, Int. J. Numer. Methods Fluids **9** (1989), 987–1009.
8. A. Karvinen, H. Ahlstedt, *Comparison of turbulence models in case of three dimensional diffuser*, M. Beaudoin, H. Jasak (eds.), *OpenSource CFD International Confference*, FSB, Berlin, 2008.
9. R. Maduta, *An eddy-resolving Reynolds stress model for unsteady flow computations: development and application*, Ph. D. thesis, Vom Fachbereich Maschinenbau an der Technischen Universität Darmstadt, 2013, 1–142.
10. D. R. Radenković, J. M. Burazer, Dj. M. Novković, *Anisotropy Analysis of Turbulent Swirl Flow*, FME Transactions **42** (2014), 19–25.
11. B. E. Launder, D. B. Spalding, *The numerical computation of turbulent flows*, Comput. Methods Appl. Mech. Eng. **3** (1974), 269–289.
12. A. S. Ćoćić, *Modeliranje i numeričke simulacije vihornih strujanja*, Doktorska disertacija, Mašinski fakultet, Univerzitet u Beogradu, 2013, 1–248.
13. B. E. Launder, G. J. Reece, W. Rodi, *Progress in the development of a Reynolds-stress turbulence closure*, Jounal of Fluid Mechanics **68** (1975), 537–566.
14. J. Lee, S. J. Jang, H. J. Sung, *Direct numerical simulations of turbulent flow in a conical diffuser*, J. Turbul. **13** (2012), 1–29.

О УТИЦАЈУ ТУРБУЛЕНТНЕ КИНЕТИЧКЕ ЕНЕРГИЈЕ НА ТАЧНОСТ $k-\varepsilon$ И LRR МОДЕЛА ТУРБУЛЕНЦИЈЕ

Резиме. Рад представља истраживање утицаја нивоа турбулентне кинетичке енергије (TKE) на тачност модела турбуленције заснованих на Рејнолдсовом осредњавању Навије-Стоксових једначина (RANS). На основу дефиниције Бусинскове хипотезе урађена је теоријска анализа утицаја нивоа турбулентне кинетичке енергије на тачност RANS модела турбуленције. Након тога, наведена теоријска анализа је истраживана упоређивањем нумерички и експериментално добијених резултата на тестном примеру стационарног безвихорног нестишљивог струјања у правом конусном дифузору, названом Азадав дифузор. Нумерички прорачуни су урађени применом OpenFOAM софтвера за рачунарску механику флуида (CFD), уз коришћење модела турбуленције првог и другог реда. Ниво турбулененте кинетичке енергије, профили брзине и Рејнолдсови напони су израчунати низсртујно у четири различита попречна пресека дифузора. Упоређивањем профила брзине, нивоа турбулентне кинетичке енергије и Рејнолдсовог напона у изабраним попречним пресецима, изведни су одређени закључци о моделирању турбулентног струјања применом $k-\varepsilon$ и LRR модела турбуленције.

(Received 01.12.2017)
(Revised 20.06.2018)
(Available online 13.07.2018)

Department of Mechanics
Faculty of Technical Sciences
University of Priština
Kosovska Mitrovica
Serbia
djordje.novkovic@pr.ac.rs

Department of Fluid Mechanics
Faculty of Mechanical Engineering
University of Belgrade
Belgrade
Serbia
jburazer@mas.bg.ac.rs

Department of Fluid Mechanics
Faculty of Mechanical Engineering
University of Belgrade
Belgrade
Serbia
acocic@mas.bg.ac.rs

Department of Fluid Mechanics
Faculty of Mechanical Engineering
University of Belgrade
Belgrade
Serbia
mlecic@mas.bg.ac.rs