Assessment of RANS to predict flows with large streamline curvature

J L Yin, D Z Wang, H Cheng and W G Gu

School of Mechanical Engineering, Shanghai Jiao Tong University, Shanghai, 200240, China

E-mail: yinjunlian@sina.com

Abstract. In order to provide a guideline for choosing turbulence models in computation of complex flows with large streamline curvature, this paper presents a comprehensive comparison investigation of different RANS models widely used in engineering to check each model’s sensitivity on the streamline curvature. First, different models including standard $k-\varepsilon$, Realizable $k-\varepsilon$, Renormalization-group (RNG) $k-\varepsilon$ model, Shear-stress transport $k-\omega$ model and non-linear eddy-viscosity model $\nu^2$-$f$ model are tested to simulate the flow in a 2D U-bend which has the standard bench mark available. The comparisons in terms of non-dimensional velocity and turbulent kinetic energy show that large differences exist among the results calculated by various models. To further validate the capability to predict flows with secondary flows, the involved models are tested in a 3D 90° bend flow. Also, the velocities are compared. As a summary, the advantages and disadvantages of each model are analysed and guidelines for choice of turbulence model are presented.

1. Introduction

Computational fluid dynamics (CFD) has emerged as a powerful hydraulic engineering design tool, such as the flows around hydraulic structure (e.g. bridge foundation scour[1; 2], open-channel flows in blood rivers[3]) and hydraulic machineries (e.g. pump and hydro turbines). Almost in all CFD applications, the choice of turbulence model is a key point, which has a great effect on the numerical results, especially in complex flows bounded in curved walls frequently countered in hydraulic engineering. The models in practical applications are almost based on RANS models due to its accuracy and economic computational cost. Such models include the standard $k-\varepsilon$ model by Launder[4], RNG $k-\varepsilon$ model by Orszag[5], Realizable $k-\varepsilon$ model by Shih[6], $k-\omega$ model by Wilcox[7], SST $k-\omega$ model by [8], $\nu^2$-$f$ model by Durbin[9]. Such models are already embedded in popular commercial CFD packages and can be easily applied to various industrial problems in an engineering sense. However, among RANS models, each eddy-viscosity based model almost has the same flaw that the modeled Reynolds stresses are isotropic. And the above mentioned models show different characteristics in practical cases. As the issues concerned in hydraulic engineering, most boundaries of the cases like rivers, coasts and hydraulic machineries are bounded by curved walls, which is supposed to produce great anisotropy in the flow structures. Curvature is present in many turbulent engineering flows, with one specific example being in turbomachinery applications. Whether it is, for example, the curvature of the blades in an axial machine, or the axial to radial transition in a centrifugal machine, curvature will exist somewhere in the system in most applications.
However, due to the lack of a compressive quantitative comparison of sensibility of turbulence models on the curved flows, there is no clear guideline for turbulence model choosing. On the other hand, although some modifications to improve the sensitivity of traditional models on the streamline curvature effects have been explored, the bases used to improve are various (e.g. standard \( k - \varepsilon \) model, SST \( k - \omega \) model, \( \nu^2-f \) model). Hence, to this end, quantitative comparisons of different two-equation eddy-viscosity models embedded in the commercial CFD package FLUENT are carried out in this paper. The numerical results are compared to typical cases with available experimental data and the different performances of each model are illustrated in terms of velocity profiles and kinetic turbulent energy.

2. Description of RANS modelling methods

The According to the Boussinesq hypothesis, Reynolds stresses \(-\overline{\rho u_i u_j}\) in Reynolds-averaged Navier-Stokes (RANS) equations can be expressed in terms of mean velocity gradients:

\[
-\overline{\rho u_i u_j} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \rho \left( \frac{\partial u_k}{\partial x_k} \right) \delta_{ij}
\]

(1)

Among the methods to model Reynolds stresses, the two-equation eddy-viscosity models generally establish two additional transport equations (for the turbulence kinetic energy, \( k \), and either the turbulence dissipation rate, \( \varepsilon \), or the specific dissipation rate, \( \omega \)) are solved, and the turbulent viscosity \( \mu_t \) is computed as a function of \( k \) and \( \varepsilon \) or and \( \omega \).

In the standard \( k - \varepsilon \) model, the turbulent viscosity \( \mu_t \) is:

\[
\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}
\]

(2)

Where \( \rho \) is fluid density, \( C_{\mu} \) is a constant. Hence, no special treatment is designed for streamline curvature effects. By means of a rigorous statistical technique (called renormalization group theory), some refinements are realized in the RNG \( k - \varepsilon \) model. It provides an option to account for the effects of swirl or rotation by modifying the turbulent viscosity. With the aid of such modification, the RNG \( k - \varepsilon \) model is expected to have a better performance in swirling flows. As another improvement of standard \( k - \varepsilon \) model, the Realizable \( k - \varepsilon \) model has many advantages. In this model, the turbulent viscosity coefficient is no longer constant but a function of mean rate-of-rotation tensor and mean strain rate tensor. Also, this model is expected to have superior performance for flows with large streamline curvature.

The \( k - \omega \) based models (e.g. standard \( k - \omega \) model, SST \( k - \omega \) model,) also compute turbulent viscosity in similar ways, but no ad hoc modifications for streamline curvature are made.

In addition, the \( \nu^2-f \) model is similar to the standard \( k - \varepsilon \) model, but incorporates near-wall turbulence anisotropy and non-local pressure-strain effects. The advantage is that no wall function is needed. However, whether it is sensible to streamline curvature effects is unknown. In this sense, this model is included in this comparison study.

3. Comparisons of RANS models in classical flow cases

To estimate the performances of the above-mentioned models, two problems with well documented experimental data available are chosen. The first one is 2D U-bend flow measured by Monson[10], which is widely used in turbulence model validation and flow passages in hydraulic machineries. The second one is 3D 90° bend flow measured by Patel[11], which is also a good benchmark for model validation for the prominent three dimensional effects induced by secondary flow. As is known from theoretical analysis and experimental measurement, the velocity of fluid particles will increase near the convex walls and decrease near the concave walls when subjected to curved walls, and the turbulent kinetic energy near the convex walls will be suppressed and that near the concave wall will be strengthened. Thus, whether one turbulence model can sensitize the streamline curvature effects
can be directly reflected on the velocity profiles and turbulent kinetic energy profiles calculated by the concerned turbulence model. Hence, the two quantities are chosen as objectives for comparison. Both cases were simulated on the platform of Fluent, which adopts finite volume method.

In addition, as an important part of turbulence modeling, enhanced wall function is adopted for standard $k-\varepsilon$ model, RNG $k-\varepsilon$ model, Realizable $k-\varepsilon$ model. Computations were run using the SIMPLEC pressure-correction solver with the QUICK convective scheme for all flow variables. The residuals for criterion of convergence were set to $10^{-6}$. Grid sensitivity study was carried out for both cases, for details, the procedure and results were described in the 2D case. For the two-dimensional case (U-bend flow), the computation was run on two processors of a Lenovo computer with 2.2 GHz and 2.0 GB RAM. The three-dimensional cases ($90^\circ$ bend and centrifugal pump) were run on 16 processors of a parallel computing cluster DeepComp 1800 with 8 CPUs×1.9 GHz and 8 GB RAM.

3.1. 2D U-bend flow

Some Two-dimensional U-bend flow is a canonical benchmark to validate a turbulence model’s sensitivity to streamline curvature. The simulation here follows the experimental geometry and conditions of Ref.[10] shown in Figure 1. The Reynolds number based on the mean velocity and channel height was $Re=10^6$. The profiles of velocity and turbulent kinetic energy used to set the inlet boundary condition $s/H=0$ ($s$ denotes streamwise coordinate, $h$ used in figures below denotes transverse coordinate, where $h/H=0$ denotes the inner wall, and $H$ is the height of the bend ) are linearly interpolated from the experimental data. The type of outflow condition is applied at the outlet.

The computational domain is displayed in Figure 2 and discretized by multi-blocked structured mesh. To get a solution independent of grid, three grid sets ($729 \times 33$, $729 \times 66$, $729 \times 100$ in the streamwise and wall normal directions, respectively) with different grid density were tested. Based on the streamwise velocity normalized by the averaged velocity at the inlet, the results shown in Figure 5 calculated from three grids are more or less the same except in the zone $0<h/H<0.1$. Thus the second set of grid ($729 \times 66$) shown was chosen for the final simulation. And similar procedures were also done for the following cases in the next section.

Figure 3 and Figure 4 show the flow development at the $90^\circ$ bend section by the streamwise velocity and turbulent kinetic energy profiles, each normalized by the average velocity $U_m$ across the channel. From the velocity distribution across the channel, flow particles will accelerate near the inner wall and decelerate near the outer wall. Although almost all the turbulence models can predict such an overall tendency, the numerical results calculated by the RNG model and RKE model agrees better with experimental data. Also, the standard $k-\omega$ model works worst among these models.

Regarding to turbulent kinetic energy, it is known that the convex curvature has a stabilizing effect on the turbulence, while the concave curvature has a destabilizing effect. The mechanism can be directly related to the velocity profiles in figure 3. The flow acceleration can induce laminarization leading to the reduction of turbulent kinetic energy, and the flow deceleration vice versa. From the turbulent kinetic energy profiles at the $90^\circ$ bend section shown in figure 4 the standard $k-\varepsilon$ model and the standard $k-\omega$ model failed to capture such a physical law. Among other models, only Realizable $k-\varepsilon$ model can predict reasonable turbulent kinetic energy profile which agrees best with the experimental data.

Figure 5 shows the profiles for $u$ at $\Theta = 180^\circ$, which corresponds to the exit of the U-bend region. Experimental data show that a separation bubble exists at this location near the inner side wall. The phenomenon of separation is always a challenge for turbulence models, which is a suitable criterion to access various models. Judging from the calculated streamwise velocity profiles, it can be seen that no negative velocity is obtained by the standard $k-\varepsilon$ model and the standard $k-\omega$ model, the profiles by Realizable $k-\varepsilon$ model and RNG $k-\varepsilon$ model are more or less the same, the SST $k-\omega$ model performs best, which shows its advantage in separated flows. Although the reverse flow is captured by the $v^2-f$ model, the velocity profile is far away from the experimental data.
Figure 6 shows the profiles for normalized kinetic energy at $\theta = 180^\circ$. From the results, the most significant difference between data and model results is found in the large peak of turbulent kinetic energy on the inner wall. York[12] hypothesized that this peak is induced by the unsteady characteristics of the separated shear layer, and that RANS models in general are ill-suited to accurately capture the phenomenon of shear layer instability and breakup. For comparison, again, the standard $k-\varepsilon$ model and the standard $k-\omega$ model show a symmetric distribution of $k$. Among other models, the Realizable $k-\varepsilon$ model shows a superior behavior than others, which suggests its good potential for further improvement.

![Figure 1. Geometry of 2D U-bend with specified boundary conditions](image1)

![Figure 2. Grid generation of 2D U-bend with refinement near the walls](image2)

![Figure 3. Normalized streamwise velocity distribution at $\theta=90^\circ$](image3)

![Figure 4. Normalized turbulent energy distribution at $\theta=90^\circ$](image4)

![Figure 5. Normalized streamwise velocity distribution at $\theta=180^\circ$](image5)

3.2. 3D 90° bend flow

The flow development in a 90° bend is very similar to those cases in engineering such as rivers and hydraulic machinery. Compared with the 2D U-bend flow, flow in 3D 90° bend is not characterized by streamline curvature but also prominent secondary flow, which is an important focus in engineering, especially in the hydraulic design of turbomachinery. In this sense, it is a useful benchmark for validation of turbulence models before being extended to applications in pumps and turbines. The experimental data for this test case have been provided by the ERCOFTAC database, and refer to the measurements of Kim and Patel using pressure probe and hot-wire measurement techniques. The data are very detailed and cover the majority of the cross-sections where major full 3D flow phenomena are developing. The advantage of the detailed measurements is that they can be used effectively as a reference for the validation of turbulence models.

The computational domain for this case is shown in Figure 7. The inlet is at a distance equal to $4.5H$ upstream of the bend and the outlet is at a distance $30H$ downstream of the bend. The span extends $6H$ in the z-direction, with the $z/H=0$ defined to correspond to the bottom walls. The Reynolds number computed using the duct height $H = 0.203$ m and the inlet velocity $U_m = 16$ m/s, is equal to 224,000. The grid dimensions shown were $172 \times 138 \times 132$, which corresponds to the finest grid tested.
in the study by Yakinthos [13]. The value of $y^+$ near all wall boundaries was controlled to be less than 10, which satisfies the requirements of the enhanced wall function in the two-layer $k-\varepsilon$ model.

To assess the response to the secondary flow induced by the curvature, we focused on the flow development at the position located at $45^\circ$ station and for $z/H=0.0625$. The three mean velocity components were chosen for comparison between simulations and experimental data. In Figures 12-14, the coordinate direction $h$ indicates the transverse dimension with $h=0$ corresponding to the convex (outer) surface and $h=1$ corresponding to the concave (inner) surface. Figure 8 shows the streamwise velocity distribution, normalized by the average velocity across the channel. The acceleration of fluid on the convex side and the deceleration on the concave side are more accurately captured by all models with little difference.

Figure 9 shows the normalized transverse velocity distribution which is defined negative (from inner side to outside). Experiments indicate that a vortex pair is formed at this location in the channel. The effect of the vortex formation is to produce negative transverse velocity over the channel height, indicating secondary flow in the direction from the convex surface towards the inner concave surface. Among these models tested, the Realizable $k-\varepsilon$ model and RNG $k-\varepsilon$ model shows this behavior most clearly, most close agreement with the experimental data. Thus, to get a relative more accurate velocity distribution, the two models are recommended to be adopted.

Finally, Figure 10 shows the normalized spanwise velocity distribution at this location in the channel. The large positive value near the convex surface is indicative of the secondary vortex motion. Agreement near the concave surface is good for all RANS models; again, the profile calculated by RNG $k-\varepsilon$ model is the closest to experimental data. However, the behaviour on the convex side ($h/H > 0.6$) is not well captured by all the models, which reason is not clear now.

4. Conclusion
In this study, a comprehensive comparison study of RANS based turbulence models widely used in engineering was carried out to test their sensibility on the streamline curvature. Judging from the computed profiles of normalized velocities and turbulence kinetic energy from the 2D U-bend flow, it can be concluded that the standard $k-\omega$ model is not so sensible for curved flow, while the SST
$k-\omega$ model has a relative better performance, especially in the more accurate prediction of separated bubble at the exit of U-bend. For $k-\varepsilon$ model series, the RNG and Realizable models has a superior performance for the prediction of velocity profiles, while Realizable $k-\varepsilon$ model can predict turbulent kinetic energy more accurately, which is great of importance for heat transfer. In addition, the $\nu^2-f$ model behaves in general for curved flow, which still need some improvement to develop its advantage which doesn’t need wall functions. As to the sensibility on the secondary flow in a typical 3D 90° bend, it is concluded that the RNG $k-\varepsilon$ model can more accurate transverse velocity, which is very useful for engineering computation such as hydraulic performance prediction in pumps and turbines. Overall, for flows bounded by curved boundaries in engineering, the RNG $k-\varepsilon$ model is recommended if the accurate profiles of velocity distribution is desired and if the quantity transport such as mass transfer and heat transfer is focused, it is recommended to employ the Realizable $k-\varepsilon$ model.

Acknowledgement
The research work was funded by China Postdoctoral Science Foundation (2013M531173), Shanghai Postdoctoral Science Foundation (13R21414100) and the National Natural Science Foundation of China (No. 51106095).

References:
[1] Ge L and Sotiropoulos F 2005 J. Hydraul Eng-Asce 131 800-8
[2] Ge L, Lee SO, Sotiropoulos F and Sturm T 2005 J. Hydraul Eng-Asce 131 809-20
[3] Nezu I and Sanjou M 2006 J. Hydraul Eng-Asce 132 681-95
[4] Launder B E and Spalding D B 1972 Lectures in Mathematical Models of Turbulence
[5] Orszag S A, Yakhot V, Flannery WS, Boysan F, Choudhury D, Maruzewski J and Patel B 1993 Renormalization group modeling and turbulence simulations Near-wall turbulent flows 1031-46
[6] Shih T H, Liou W W, Shabbir A, Yang Z and Zhu J 1995 Computers & Fluids 24 227-38
[7] Wilcox D C 1998 Turbulence Modeling for CFD (California: DCW Industries, Inc)
[8] Menter F R 1994 AIAA Journal 32 1598-605
[9] Durbin P A 1995 AIAA Journal 33 659-64
[10] Monson B J, Seegmiller H L and McConnaughey P K 1989 Comparison of LDV measurements and Navier-Stokes solutions in a two-dimensional 180-degree turn-around duct AIAA 89-275.
[11] Kim W J and Patel VC 1993 An experimental study of boundary-layer flow in a curved rectangular duct. In Symposium on Data for Validation of CFD codes ASME Fluids Engineering Division (1993, Washing DC) 13-28
[12] York W D, Walters D K and James H L 2009 Int. J. Numerical Method for Hwat & Fluid Flow 19 745-76
[13] Yakinthos K, Vlahostergios Z and Goulas A 2008 Int. J. Heat and Fluid Flow 29 35-47