PAPER • OPEN ACCESS

Comparison of Various Turbulence Models for Unsteady Flow around a Finite Circular Cylinder at *Re*=20000

To cite this article: Di Zhang 2017 J. Phys.: Conf. Ser. 910 012027

View the article online for updates and enhancements.

Related content

- <u>Turbulence modeling for Francis turbine</u> water passages simulation P Maruzewski, H Hayashi, C Munch et al.
- <u>Simulation of flow in a simplified draft tube:</u> <u>turbulence closure considerations</u> H Foroutan and S Yavuzkurt
- <u>Numerical prediction of air flow within</u> street canyons based on different twoequation *k*-models
 Afiq Witri Muhammad Yazid, Nor Azwadi

Che Sidik, Salim Mohamed Salim et al.

IOP Conf. Series: Journal of Physics: Conf. Series 910 (2017) 012027

Comparison of Various Turbulence Models for Unsteady Flow around a Finite Circular Cylinder at *Re*=20000

Di Zhang

School of Civil, Environmental & Mining Engineering, The University of Western Australia, 35 Stirling Highway, Crawley, WA 6009, Australia. Email: di.zhang@uwa.edu.au.

Abstract. This paper compares the performance of eight Reynolds-Averaged Navier–Stokes (*RANS*) two-equation turbulence models and two sub-grid scale (*SGS*) large eddy simulation (*LES*) models in the scenario of unsteady flow around a finite circular cylinder at an aspect ratio (*AR*) of 1.0 and a Reynolds number of *Re*=20000. It is found that, among all the eight *RANS* turbulence models considered, the *K-Omega-SST* model (*viz. SST-V2003*) developed by Menter *et al.*^[1,2] possesses the best overall performance (being closest to the numerical results of the two *LES* models considered, which can be deemed as the quasi-exact solution in view of the very fine computational mesh employed by the two *LES* models in this study) in terms of the mean surface pressure coefficient distribution (*i.e.* C_p), the mean drag coefficient (*i.e.* C_d), the mean streamline profiles in some characteristic planes (such as the mid-height plane and the symmetry plane of the cylinder) and the distribution of mean bed-shear-stress amplification on the bottom wall.

1. Introduction

The experimental and numerical study on the complex three-dimensional flow structures around a bluff body remains one of the most active areas of research in fundamental fluid dynamics over the past decades, mainly due to the extensive presence of such flows in nature and engineering applications, such as the wind field around high-rise buildings, the pollutant transport around chimney stacks, the aerodynamics force on cooling towers, the flow field around offshore structures, the heat exchange on electronic circuit boards, and so on ^[3-6]. Although many earlier studies focused on the analysis of the flow past a nominally infinite two-dimensional (2D) circular or square cylinder, recently most attentions have been paid to the unsteady flow around finite-height cylinders ^[7-12], with one end immersed in the free stream (*viz.* the free end) and the other end mounted on a flat wall (*viz.* the base end), which are more consistent with the structures in reality. Correspondingly, due to the combined influence of the downwash flow from the free end and the boundary layer near the bottom wall, the three-dimensional (3D) flow structure around a finite cylinder is usually much more complicated than that behind an infinite one.

It can be concluded from the existing literature that all the following six factors can have some effects on the flow structure around a finite-height cylinder ^[6-8]: 1). the turbulence intensity of the approaching flow; 2). the cross-section shape of the cylinder; 3). the *Re* number (*viz. Re=UD/v*, where *D* is the characteristic width of the cylinder, *U* is the free stream velocity, and *v* is the fluid's kinematic viscosity.); 4). the boundary-layer thickness on the bottom wall relative to the cylinder height (*viz.* δ/h); 5). the ratio of the cylinder height to the characteristic width of the cylinder (*viz.* AR=h/D); 6). the blockage ratio of the channel (*viz.* $\beta_1=h/H$ and $\beta_2=D/B$, where *H* and *B* are the height and width of the channel or the computational domain, respectively.). On one hand, the respective effect extent of the

CTCE2017	IOP Publishing
IOP Conf. Series: Journal of Physics: Conf. Series 910 (2017) 012027	doi:10.1088/1742-6596/910/1/012027

aforementioned factors may vary significantly from each other when it comes to a specific condition. On the other hand, in view of multiple influencing factors and complex flow field under this circumstance, in the past, researchers often only discuss one or two factors' influence in each article for simplifying the problem, therefore, the integrated effect of simultaneously changing several parameters remain to be investigated in the future.

Considering that high Reynolds numbers always result in very complicated vortex structures in the wake of a finite circular cylinder (especially when mounted on a non-slip bottom wall), this paper presents a detailed discussion on the flow around a finite circular cylinder with AR=1 at a relatively high *Re* numbers (*viz.* 20000). The purpose of this study is to quantitatively and qualitatively make a detailed comparison of different turbulence models when it comes to the mean pressure coefficient profile C_p in the mid-height plane of the cylinder, the mean drag coefficient C_d of the cylinder, the time-averaged velocity and pressure fields, and the distribution of mean bed-shear-stress amplification on the bottom wall.

2. Configuration and Numerical Model

2.1 Test Configuration

A similar geometric configuration as that used by Zhang *et al.* ^[6, 8] is employed in this study. As illustrated in *Fig.* 1, a finite circular cylinder with a non-dimensional width of D=1 and a non-dimensional height of h=1 is vertically mounted on a plane boundary, and the (streamwise) length, (transverse) width and (spanwise) height of the computational domain are, respectively, L=30D, W=22D and H=2D (which gives an area blockage ratio of 2.27%). Further, the junction section between the cylinder and the bottom wall is centered at the origin of the coordinate system, which means that the inlet boundary is located at 10D upstream of the cylinder and the outlet boundary is situated at 20D downstream of the cylinder (*i.e.* $L_1=10D$, $L_2=20D$).

2.2 Governing Equations

2.2.1. RANS models. The unsteady 3D Reynolds-averaged Navier-Stokes (RANS) equations employed in this study can be obtained by taking ensemble average of instantaneous mass and momentum conservation equations for incompressible and isothermal flows, as shown in the following ^[6-8]:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\overline{u}_i \overline{u}_j + \overline{u'_i u'_j}) = -\frac{1}{\rho} \cdot \frac{\partial \overline{p}}{\partial x_i} + \frac{\partial \overline{\tau}_{ij}}{\partial x_j}.$$
(2)

where the subscripts *i* and *j* indicate the *i*th and *j*th components of the Cartesian coordinate respectively, \overline{u} and \overline{p} represent the time-averaged velocity and pressure fields respectively, *t* is the time, ρ is the density of the fluid (defined as a constant in this study, *viz.* ρ =1000), $\overline{\tau}_{ij} = 2vS_{ij} = v(\partial \overline{u}_i/\partial x_j + \partial \overline{u}_j/\partial x_i)$ is the mean viscous stress tensor, $S_{ij} = (\partial \overline{u}_i/\partial x_j + \partial \overline{u}_j/\partial x_i)/2$ is the mean strain-rate tensor, and *v* is the kinematic viscosity of the fluid (*i.e.* $v=5\times10^{-5}$, which leads to a Reynolds number of $Re=2\times10^4$). Obviously, the Reynolds stress tensor term (*i.e.* $\overline{T}_{ij} = \overline{u'_iu'_j}$) can be obtained by using the Boussinesq eddy-viscosity hypothesis:

$$\overline{T}_{ij} = \overline{u'_i u'_j} = \frac{2}{3} k \delta_{ij} - v_t \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right)$$
(3)

$$k = \frac{1}{2} \overline{u'_{i}u'_{i}} = \frac{1}{2} \left(\overline{u'_{x}u'_{x}} + \overline{u'_{y}u'_{y}} + \overline{u'_{z}u'_{z}} \right)$$
(4)

where k is the turbulent kinetic energy, δ_{ij} is the Kronecker delta symbol and v_t represents the turbulent viscosity of the flow. In order to obtain the turbulent viscosity v_t , different RANS turbulent models are adopted and compared with each other here, including the standard k-epsilon $(k-\varepsilon)$ model ^[13], the renormalization group k-epsilon (RNG $k-\varepsilon$) model ^[14], the realizable $k-\varepsilon$ turbulence model ^[15], Lien's cubic non-linear low-Reynolds $k-\varepsilon$ model ^[16], Launder and Sharma low-Reynolds $k-\varepsilon$ turbulence model ^[17], Shih's quadratic non-linear $k-\varepsilon$ turbulence model ^[18], the standard high Reynolds-number k-omega $(k-\omega)$ model ^[19] and the K-Omega–SST $(k-\omega-SST)$ turbulence model ^[1, 2]. For the conciseness of the presentation, the details of these models are not presented here, readers can refer to *Ref.* [1, 2, 13-19] for further details.

2.2.2. LES models. The spatial filtering of unsteady 3D instantaneous mass and momentum equations in a Newtonian incompressible flow results in the following LES equations ^[20-23]:

$$\frac{\partial \tilde{u}_i}{\partial x_i} = 0.$$
⁽⁵⁾

$$\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\tilde{u}_i \tilde{u}_j) = -\frac{1}{\rho} \cdot \frac{\partial \tilde{p}}{\partial x_i} + 2\partial_j \left(\nu \tilde{S}_{ij} \right) - \partial_j \left(\tilde{\tau}_{ij} \right).$$
(6)

$$\tilde{S}_{ij} = \frac{1}{2} \left(\frac{\partial \tilde{u}_j}{\partial x_i} + \frac{\partial \tilde{u}_i}{\partial x_j} \right), \quad \tilde{\tau}_{ij} = (\tilde{u_i u_j} - \tilde{u}_i \tilde{u}_j).$$
(7)

where \tilde{u} and \tilde{p} are the filtered velocity and pressure fields respectively, \tilde{S}_{ij} is the filtered or resolved scale strain-rate tensor, and $\tilde{\tau}_{ij}$ is the unknown SGS stress tensor, representing the effects of the SGS motions on the resolved fields of LES, which needs to be modelled by using a so-called SGS model, for the purpose of closing the above governing equations. By employing the Boussinesq eddy-viscosity hypothesis, the SGS stress tensor can be expressed as:

$$\tilde{\tau}_{ij} = v_{sgs} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) + \frac{1}{3} \delta_{ij} \tilde{\tau}_{il} \,. \tag{8}$$

For the Smagorinsky SGS model ^[20, 21]:

$$v_{sgs} = \left(C_{S}\tilde{\Delta}\right)^{2}S, \quad S = \left(2\tilde{S}_{ij}\tilde{S}_{ij}\right)^{\frac{1}{2}}, \quad \tilde{\Delta} = (\Delta x \Delta y \Delta z)^{\frac{1}{3}}.$$
(9)

For the One-Equation eddy viscosity model ^[22, 23]:

$$\frac{\delta k_{sgs}}{\delta t} + \nabla \cdot \left(k_{sgs}\vec{U}\right) - \nabla \cdot \left(\left(v_{sgs} + v\right) \cdot \nabla k\right) = -D_{ij} : B_{ij} - C_E \cdot k^{\frac{3}{2}} / \tilde{\Delta}$$

$$v_t = C_k \tilde{\Delta} \sqrt{k_{sgs}}, \quad D_{ij} = symm \left(\nabla \vec{U}\right), \quad B_{ij} = \frac{2}{3} K_{sgs} I_{ij} - 2v_{sgs} dev(D_{ij}). \quad (10)$$

where v_{sgs} and k_{sgs} are the sub-grid turbulent viscosity and the sub-grid turbulent kinetic energy respectively, $C_E = 1.048$, $C_k = 0.094$ and $C_s = 0.1$ are the model parameters, and $\tilde{\Delta}$ represents the local grid-resolution quality.

2.3. Boundary Conditions and Numerical Schemes

Four kinds of boundary conditions are involved:

1. Inlet: For the velocity field, a fixed uniform velocity is prescribed (*i.e.* $u_I = 1$, $v_I = w_I = 0$), and, for the pressure field, the zero-gradient condition is imposed (*i.e.* $\partial p_I / \partial n = 0$).

2. Outlet: For the velocity field, the convective outflow boundary condition is adopted:

$$\frac{\partial \phi}{\partial t} + \vec{u}_c \frac{\partial \phi}{\partial n} = 0.$$
 (11)

where ϕ denotes all the three velocity components (*viz. u, v* and *w*), and \vec{u}_c is the advective velocity at the outlet boundary. For the pressure field, the homogeneous Dirichlet condition is utilized at the outlet boundary (*i.e.* p=0).

3. Bottom wall and the surface of the obstacle: No-slip impermeable boundary condition is prescribed for the velocity field (*i.e.* u=v=w=0), and the zero-gradient condition is employed for the pressure field.



Figure.1. Schematic of the computational domain and the *Level-1* mesh employed by different RANS models.

4. Top and lateral boundaries of the computational domain: Free-slip condition is prescribed, which means that the velocity component normal to the boundary is zero (*i.e.* $u_{b\perp} = 0$) and the normal

gradients of both the pressure and the tangential velocity component are zero (*i.e.* $\partial u_{b//}/\partial n = \partial p_b/\partial n = 0$, where $u_{b//}$ is the tangential velocity component).

The numerical simulations were carried out by using the Open Source Field Operation and Manipulation (OpenFOAM) C++ libraries. Specifically, PisoFoam, one of the standard solvers provided by OpenFOAM, is selected to solve the aforementioned governing equations. The Pressure Implicit with Splitting of Operators (PISO) ^[24] algorithm is employed to deal with the pressure-velocity coupling on a collocated grid system in the context of finite volume method (FVM). In order to satisfy the boundedness property, the *limitedLinear* TVD/NVD scheme is employed to discretize the convection terms of the scalar transport equations, and the *limitedLinearV* scheme, which stands for the improved version of the *limitedLinear* TVD/NVD scheme for vector fields and whose limiter is formulated to take into account the direction of the field, is adopted to discretize the convection terms of the pressure gradient term when discretizing the governing equations, and the second-order Crank–Nicolson method is adopted for the temporal discretization.

3. Results and Discussion

3.1. Mean Drag Coefficient and Surface-Pressure Profile

Considering that in normal conditions LES turbulence models have a higher demand on the resolution of the computational mesh relative to various RANS turbulence models, two levels of grids are used in this study, namely *Level-1* for eight RANS models and *Level-2* for two LES models. As shown in the *Table-1*, for the *Level-1* mesh, it totally consists of approximately 3.26 million grid points, the circumference of the cylinder is evenly divided into 200 parts, and the near-wall grid size (defined as the distance between the centroid of the first cell and the non-slip boundary) is about 0.0016, which leads to a distance in wall units of less than 1.0 (*viz.* $(\Delta y)_{max}^+ < 1.0$). However, for the finer *Level-2* mesh, it totally consists of approximately 10.3 million grid points, the circumference of the cylinder is evenly divided into 200 parts, and the near-wall grid size is about 0.0005, which leads to a distance in wall units of less than 0.5 (*viz.* $(\Delta y)_{max}^+ < 0.5$). In addition, the time step is fixed as Δt =0.0002, which can ensure that the maximum Courant–Friedrichs–Lewy (CFL) number in all cases is not larger than 0.3 in order to improve both the temporal accuracy and the numerical stability.

In view of a much finer grid resolution is employed by the two LES models and the C_d values of two LES models are well consistent with each other (*i.e.* C_d =0.802 for the one-equation eddy viscosity LES model and C_d =0.820 for the Smagorinsky LES model), it is reasonable to assume that the C_d values obtained by adopting the two LES models can be (at least to some degree) treated as the exact value of the time-averaged drag coefficient (*i.e.* $C_{d-exaci}\approx0.80$). From *Table-1*, it can be concluded that the Standard K-Omega Model (1998) and the K-Omega–SST Model (2003) possess the best performance when evaluating the C_d value (*i.e.* C_d =0.766 and C_d =0.761, respectively), when compared with the other six RANS turbulence models, namely Shih's non-linear RSA k- ε Model (C_d =0.703), Lien-Cubic Low-Re k- ε Model (C_d =0.678), Realizable k- ε Model (C_d =0.668), Launder-Sharma Low-Re k- ε Model (C_d =0.631), Standard k- ε Model (C_d =0.625) and RNG k- ε Model (C_d =0.612).

Fig. 2 presents a quantitative comparison of the distribution of the mean surface-pressure coefficient (C_p) along the circumference at the mid-height of the cylinder (*i.e.* Z/D=0.5). Actually, the numerical results of two existing experimental studies have also been added for comparison, namely Okamoto and Sunabashiri^[7] (AR=1, $\delta/D<0.11$, $Re_D=2.5\times10^4\sim4.7\times10^4$, $H/D\approx7$) and Kawamura *et al.*^[5] (AR=1, $\delta/D=0.10$, $Re_D=3.2\times10^4$, H/D=15). It is clear from *Fig.* 2 that, even for the time-averaged surface-pressure profile at the mid-span of the cylinder, relative to the remaining six RANS turbulence models, the Standard *K-Omega* Model and the *K-Omega–SST* Model possess the best prediction accuracy as well (sharing the best consistency with the prediction results of the two LES model). Concretely, in the range of $\theta=[50^{\circ}, 120^{\circ}]$, the mean pressure profile of the *K-Omega–SST* Model is closest to that of the two LES model, but in the range of $\theta=[120^{\circ}, 180^{\circ}]$, the mean pressure profile of the *K-Omega* Model is closest to that of the two LES model.

IOP Conf. Series: Journal of Physics: Conf. Series 910 (2017) 012027

doi:10.1088/1742-6596/910/1/012027

From *Table-1* and *Fig. 2*, it can be concluded that, in terms of both the mean drag coefficient and the mean surface-pressure distribution at the mid-height of the cylinder, the Standard *K-Omega* Model (1998) and the *K-Omega–SST* Model (2003) can result in a better prediction accuracy relative to the other six RANS turbulence models considered (at least for the simulation case carried out in this study). Therefore, in the following two sections when analyzing the other two aspects of the low field, only the numerical results of the aforementioned two RANS models are presented and compared with that of the two LES models considered.

3.2. Mean Velocity and Pressure Fields

In this section, the mean velocity and pressure fields are investigated by examining the time-averaged streamlines and pressure contours in two characteristic planes (*i.e.* the mid-height plane of the cylinder (Z/D=0.5) and the symmetry plane (Y/D=0)).

Fig. 3 indicates that, in the mid-height plane of the cylinder, a strong positive/negative pressure region can be generated before/behind the cylinder because of the obstruction effect of the circular cylinder. Moreover, it is clear that the same topology can be observed for the two LES model and the *K-Omega–SST* model, in terms of the mean streamlines in this plane, containing two symmetrically distributed spiral centers (*i.e. C* and *D*) and one saddle node (*i.e. R* in *Fig.* 3). Nevertheless, when it comes to the standard *K-Omega* model, differences exist in terms of the overall topology of the time-averaged streamlines in the *Z/D*=0.5 plane when compared with that of the other three turbulence models, as verified by *Fig.* 3(*d*). The *X*-location of *R* is found to be with only slight difference for all the aforementioned four turbulence models, in view of that X(R)=1.524 for the Smagorinsky LES model, X(R)=1.504 for the one-equation eddy viscosity LES model, X(R)=1.22 for the *K-Omega–SST* model and X(R)=1.56 for the standard *K-Omega* model.

Fig. 4 presents the time-averaged streamlines and pressure contours in the symmetry plane, and the same color scale is used for all the cases for the convenience of comparison. Obviously, the overall topology is almost the same for all the four turbulence models presented in this figure, except for the fact that the standard $k-\omega$ model fails to predict the time-averaged horse-shoe vortices and the corresponding separation points in front of the cylinder, as confirmed by Fig. 4(d). In the near wake behind the cylinder, one clockwise vortex core A (caused by the downwash flow and located in the upper region) and another anti-clockwise vortex core B (induced by the very weak upwash flow close to the bottom wall and located in the lower region) coexist in the symmetry plane for all the four turbulence models. Besides, the flow separating from the leading edge of the cylinder tip reattaches onto the free-end surface, and consequently an isolated secondary recirculation region (its center is denoted by the capital letter E in Fig. 4) is formed above the tip end for all the four turbulence models. It should be emphasized that the flow field above the free end is of significant importance since the curvature of the streamlines within this region is directly related to the angle of the downwash flow behind the cylinder. Additionally, Fig. 4(a, b, c) illustrates that two mean horseshoe-vortex cores can be identified in the symmetry plane for the two LES models and the $k-\omega$ -SST model, marked by H1 and H2. But, no such vortex phenomena can be captured in Fig. 4(d) for the standard $k-\omega$ model.

From *Figs.* 3-4, it can be concluded that, in terms of the mean streamlines and pressure contours in both the mid-height plane of the cylinder and the symmetry plane, the *K-Omega-SST* model (employing a relative coarse mesh) will have a better consistency with that of the two LES turbulence models considered (employing a much finer mesh) and therefore is superior to the standard *K-Omega* model (using the same coarse mesh as the *K-Omega-SST* model).

IOP Conf. Series: Journal of Physics: Conf. Series **910** (2017) 012027

doi:10.1088/1742-6596/910/1/012027

Turbulence Model	Grid Resolution	Number of Nodes	Node-Number Along the Circumference	Near-wall Grid Dimension	Near-wall Grid Y^+	Mean Drag Coefficient C_d
Standard <i>K-Omega</i> Model (1998)	Level-1	3.26 Million	200	0.0016	1	0.766
<i>K-Omega–SST</i> Model (2003)	Level-1	3.26 Million	200	0.0016	1	0.761
Shih's non-linear RSA <i>k-ε</i> Model (1993)	Level-1	3.26 Million	200	0.0016	1	0.703
Lien-Cubic Low-Re k- ε Model (1996)	Level-1	3.26 Million	200	0.0016	1	0.678
Realizable <i>k</i> -ε Model (1995)	Level-1	3.26 Million	200	0.0016	1	0.668
Launder-Sharma Low- Re <i>k-ε</i> Model (1974)	Level-1	3.26 Million	200	0.0016	1	0.631
Standard k - ε Model (1972)	Level-1	3.26 Million	200	0.0016	1	0.625
RNG <i>k</i> -ε Model (1992)	Level-1	3.26 Million	200	0.0016	1	0.612
One-Equation LES Model (1986)	Level-2	10.3 Million	248	0.0008	0.5	0.802
Smagorinsky LES Model (1963)	Level-2	10.3 Million	248	0.0008	0.5	0.820

Table.1. Gr	rid resolution	and the mean	drag	coefficient	C_{d} .
					- u ·



Figure. 2. Comparison of the mean surface-pressure coefficient along the circumference at Z/D=0.5.

doi:10.1088/1742-6596/910/1/012027

IOP Conf. Series: Journal of Physics: Conf. Series 910 (2017) 012027



Figure. 3. Comparison of the mean streamlines and pressure contours on the Z/D=0.5 plane.



Figure. 4. Comparison of the mean streamlines and pressure contours on the Y/D=0 plane.

3.3. Bed-Shear-Stress Distribution

This section aims to investigate the distribution of the time-averaged bed shear stress and further give the full picture of the bed-shear-stress amplification (based on the velocity gradient at the height of Z/D=0.0001) around the cylinder for different turbulence models. *Fig.* 5 demonstrates a comparison of the time-averaged bed-shear-stress amplification along the symmetry line at the upstream side of the cylinder. It is evident that the both the *K-Omega-SST* model and the standard *K-Omega* model possess

the same main features (namely the overall shape) as the experimental and numerical results of Roulund *et al.* ^[9] ($|\tau_{max}|\approx 2.0$, AR=1, $\delta/D=1$, $\delta/H=1$, $Re_D=1.7\times10^5$) and *Ming et al.* ^[8] ($|\tau_{max}|\approx 1.45$, AR=1, $\delta/D=1$, $\delta/H=1$, $Re_D=1.7\times10^5$). However, when it comes to the location and value of the most negative amplification factor, the *K-Omega-SST* model results in a value of $|\tau_{max}|\approx 1.3$, which is slightly more accurate than that of the standard *K-Omega* model ($|\tau_{max}|\approx 1.1$). Furthermore, the Smagorinsky LES model and the one-equation eddy viscosity LES model will lead to a value of $|\tau_{max}|\approx 1.8$ and $|\tau_{max}|\approx 2.2$ respectively, which are almost identical to the experimental results of Roulund *et al.* ^[9] ($|\tau_{max}|\approx 2.0$) due to the much finer computational mesh adopted by the two LES models. Additionally, *Fig.* 6 compares the full picture of the mean bed-shear-stress amplification around the cylinder for different turbulence models. Two symmetrically distributed points with the largest bed shear stress can be identified at an angle of $\theta=[55^\circ, 60^\circ]$ (θ represents the angle measured from the negative *X* axis) for each test case, being consistent with the conclusions of Roulund *et al.* ^[9] $\theta=[45^\circ, 70^\circ]$. Obviously, no significant difference can be obtained in terms of the maximum bed-shear-stress amplification at these locations



Figure. 5. Comparison of the bed-shear-stress along the symmetry line in front of the cylinder.

doi:10.1088/1742-6596/910/1/012027

IOP Conf. Series: Journal of Physics: Conf. Series 910 (2017) 012027



Figure. 6. Comparison of the full picture of the mean bed-shear-stress on the bottom wall.

for different turbulence models, in consideration of that $|\tau_{\text{max-lateral}}|\approx 3.86$ for the Smagorinsky LES model, $|\tau_{\text{max-lateral}}|\approx 4.34$ for the One-Equation LES model, $|\tau_{\text{max-lateral}}|\approx 3.42$ for the *K-Omega-SST* model and $|\tau_{\text{max-lateral}}|\approx 3.67$ for the standard *K-Omega* model.

4. Conclusions

Turbulent flow past a finite circular cylinder (AR=1) is simulated at a relatively large Reynolds number (Re=20000). The focus is to examine the influence of different turbulence models on several aspects of the flow, namely the mean drag coefficient, the mean surface-pressure coefficient, the timeaveraged velocity and pressure fields, and the mean bed-shear-stress amplification. It can be concluded that, in terms of all the aforementioned aspects, the numerical results of the K-Omega-SST model (employing a relative coarse mesh) will have a better consistency with that of the two LES turbulence models considered (employing a much finer mesh) and therefore is more accurate than the standard K-Omega model and the other six remaining RANS turbulence models (using the same coarse mesh as the K-Omega-SST model). Therefore, the superiority of the K-Omega-SST model (viz. SST-V2003, developed by Menter *et al.*^[11]) has been effectively validated for the present numerical case and this turbulence model is highly recommended in the scenario of the adverse pressure gradients and separating flow around a finite cylinder.

5. Acknowledgments

The author wish to acknowledge the financial support provided by Australian Research Council Discovery Grant (Project ID: DP150104644) and the computational resources provided by the Pawsey Supercomputing Centre with funding from the Australian Government and the Government of Western Australia.

6. References

- [1] F.R. Menter, M. Kuntz, R. Langtry. "Ten years of industrial experience with the SST turbulence model". *Turbulence, Heat and Mass Transfer*, **4**(1), 625-632 (2003).
- [2] F.R. Menter. "Two-equation eddy-viscosity turbulence models for engineering applications". *AIAA Journal*, **32**(8), 1598-1605 (1994).
- [3] C.H.K. Williamson, "Vortex dynamics in the cylinder wake," *Annual Review of Fluid Mechanics*. **28**(1), 477-539 (1996a).
- [4] D. Sumner, J.L. Heseltine. "Tip vortex structure for a circular cylinder with a free end". *Journal* of Wind Engineering and Industrial Aerodynamics, **96**(6), 1185-1196 (2008).
- [5] T. Kawamura, M. Hiwada, T. Hibino, I. Mabuchi, M. Kumada. "Flow around a finite circular cylinder on a flat plate: Cylinder height greater than turbulent boundary layer thickness". *Bulletin of JSME*, **27**(232), 2142-2151 (1984).
- [6] D. Zhang, L. Cheng, H. An, M. Zhao. "Direct numerical simulation of flow around a surfacemounted finite square cylinder at low Reynolds numbers". *Physics of Fluids*, 29(4), 045101 (2017).
- [7] S. Okamoto, Y. Sunabashiri. "Vortex shedding from a circular cylinder of finite length placed on a ground plane". *Journal of Fluids Engineering*, **114**(4), 512-521 (1992).
- [8] D. Zhang, L. Cheng, H. An, S. Draper. "Flow around a surface-mounted finite circular cylinder completely submerged within the bottom boundary layer". *Submitted to Physics of Fluids*.
- [9] A. Roulund, B.M. Sumer, J. Fredsøe, J. Michelsen. "Numerical and experimental investigation of flow and scour around a circular pile". *Journal of Fluid Mechanics*, **534**, 351-401 (2005).
- [10] O. Lehmkuhl, I. Rodríguez, R. Borrell, J. Chiva, A. Oliva. "Unsteady forces on a circular cylinder at critical Reynolds numbers". *Physics of Fluids*, **26**(12), 125110 (2014).
- [11] J.A. Bourgeois, P. Sattari, R.J. Martinuzzi. "Alternating half-loop shedding in the turbulent wake of a finite surface-mounted square cylinder with a thin boundary layer". *Physics of Fluids*, **23**(9), 095101 (2011).
- [12] D. Sumner, N. Rostamy, D.J. Bergstrom, J.D. Bugg. "Influence of aspect ratio on the mean flow field of a surface-mounted finite-height square prism". *International Journal of Heat and Fluid Flow*, **65**, 1-20 (2017).
- [13] B.E. Launder, D.B. Spalding. "The numerical computation of turbulent flows". *Computer Methods in Applied Mechanics and Engineering*, **3**(2), 269-289 (1974).
- [14] V. Yakhot, S.A. Orszag, S. Thangam, T.B. Gatski, C.G. Speziale. "Development of turbulence models for shear flows by a double expansion technique". *Physics of Fluids A: Fluid Dynamics*, 4(7), 1510-1520 (1992).
- [15] T.H. Shih, W.W. Liou, A. Shabbir, Z. Yang, J. Zhu. "A new k-ε eddy viscosity model for high reynolds number turbulent flows". *Computers & Fluids*, **24**(3), 227-238 (1995).
- [16] F.S. Lien. "Low-Reynolds-number eddy-viscosity modelling based on non-linear stressstrain/vorticity relations". *Engineering Turbulence Modelling and Measurements*, 3, 91-100 (1996).
- [17] B.E. Launder, B.I. Sharma. "Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc". *Letters in Heat and Mass Transfer*, **1**(2), 131-137 (1974).
- [18] T.H. Shih, J. Zhu, J. Lumley. "A Realizable Reynolds Stress Algebraic Equation Model". NASA Technical Memorandum, 105993, (1993).
- [19] D.C. Wilcox. "Turbulence modeling for CFD (Vol. 2, pp. 103-217)". La Canada, CA: DCW Industries (1998).
- [20] Y. Zhiyin. "Large-eddy simulation: Past, present and the future". *Chinese Journal of Aeronautics*, 28(1), 11-24 (2015).
- [21] J. Smagorinsky. "General circulation experiments with the primitive equations: I. The basic experiment". *Monthly Weather Review*, **91**(3), 99-164 (1963).
- [22] A. Yoshizawa. "Statistical theory for compressible turbulent shear flows, with the application to subgrid modeling". *The Physics of fluids*, **29**(7), 2152-2164 (1986).

IOP Conf. Se	ries: Journal	of Physics:	Conf. Series 910	(2017) 012027	doi:10.1088/1742	-6596/910/1/012027
--------------	---------------	-------------	------------------	---------------	------------------	--------------------

- [23] W. Rodi. "Comparison of LES and RANS calculations of the flow around bluff bodies". *Journal* of Wind Engineering and Industrial Aerodynamics, 69, 55-75 (1997).
- [24] O. Ubbink. "Numerical prediction of two fluid systems with sharp interfaces". Doctoral dissertation, University of London, (1997).
- [25] D. Zhang, C. Jiang, D. Liang, Z. Chen, Y. Yang, Y. Shi. "A refined volume-of-fluid algorithm for capturing sharp fluid interfaces on arbitrary meshes". *Journal of Computational Physics*, 274, 709-736 (2014).
- [26] D. Zhang, C. Jiang, D. Liang, L. Cheng. "A review on TVD schemes and a refined flux-limiter for steady-state calculations". *Journal of Computational Physics*, **302**, 114-154 (2015).
- [27] D. Zhang, C. Jiang, C. Yang, Y. Yang. "Assessment of different reconstruction techniques for implementing the NVSF schemes on unstructured meshes". *International Journal for Numerical Methods in Fluids*, 74(3), 189-221 (2014).
- [28] D. Zhang, C. Jiang, L. Cheng, D. Liang. "A refined *r*-factor algorithm for TVD schemes on arbitrary unstructured meshes". *International Journal for Numerical Methods in Fluids*, **80**(2), 105-139 (2016).