Escola de Primavera de Transição e Turbulência
Blumenau 2020

12th Spring School on Transition and Turbulence
September 21st-25th, 2020, Blumenau, SC, Brazil

EPTT-2020-0077
EVALUATION OF AN ANISOTROPIC TURBULENCE MODEL FOR SIMULATION OF THREE-DIMENSIONAL CASES

Leonardo M. Rosa
Daniela M. Koerich
Tommaso Oggian
Henry F. Meier
University of Blumenau
leorosa@gmail.com

Abstract. *Turbulence is a flow regime that can be observed in the most diverse situations. However, even though it is commonly found in nature and in industrial applications, its understanding is still challenging due to the fact that it involves a broad spectrum of scales. There are different ways to calculate the Reynolds tensor. In the* Reynolds Stress Model *(RSM), each component of this tensor is obtained by solving a separate transport equation, which implies greater computational cost when compared to simpler models such as those based on the Boussinesq hypothesis, which postulates that there is a linear relationship between the tensor itself and the strain tensor. The present study evaluates an anisotropic model to calculate the turbulent tensor. In the simulation of the cyclonic separator, all components associated with the tangential direction were obtained through the mixture length model, whereas the others through the k-ϵ model to guarantee the Galilean invariance of the Reynolds stress. This approach was implemented as a library in an open source CFD code. Tests were performed using an axisymmetric representation of a cyclone, adopting a transient formulation for the solution of the single-phase transport equations. The hybrid model predicted values closer to the experimental ones for the tangential velocity when compared against those obtained using the standard k-ϵ model, at similar computational cost. The hybrid model was also used to simulate the turbulent flow in a three-dimensional representation of the cyclone, hence demonstrating the applicability of the model to complex geometries.*

Keywords: *Modeling and Simulation, Turbulence, k-ϵ model, mixing length model, Reynolds tensor, OpenFOAM*

1. INTRODUCTION

Turbulence is a flow regime that can be observed in the most diverse situations. However, even though it is commonly found in nature and in industrial applications, its understanding is still challenging due to the fact that it involves a broad spectrum of scales. There are several approaches and models, with different levels of accuracy, that try to predict its evolution. In the Direct Numerical Simulation (DNS) approach, a highly refined mesh is generated in order to capture all flow structures. In turn, the Large Eddy Simulations (LES) requires a coarser mesh, but still sufficiently refined for the resolution of the largest scales. The smallest scales, for instance, are modeled. On the other hand, in the RANS approach, all scales of turbulence are modeled, which allows a relatively quick computation of turbulent cases. This technique applies the Reynolds decomposition to the Navier-Stokes equations and then takes an average of the resulting terms. In the momentum equation, this procedure yields the Reynolds tensor ($\tau^{\mathbf{Re}}$), which includes the turbulent fluctuations.

There are different ways to calculate the Reynolds tensor. In the *Reynolds Stress Model* (RSM), each component of this tensor is obtained by solving a separate transport equation, which implies greater computational cost when compared to simpler models. An alternative approach makes use of the Boussinesq hypothesis, which postulates that there is a linear relationship between the tensor itself and the strain tensor ($\mathbf{D}$) through a coefficient commonly called 'turbulent viscosity' (μ^t). With this approach, it is possible to group the Reynolds tensor to the diffusive term of the Navier-Stokes equation, which then considers an effective viscosity (μ^{eff}) that is the sum of the real viscosity and of the turbulent contributions ($\mu^{eff} = \mu + \mu^t$).

One of the simplest approaches to estimate the turbulent viscosity is the algebraic relationship provided by the Prandtl mixing model. However, standard two equation turbulence models are the most used in industries to perform CFD simulations, since they are robust and provide a good cost benefit ratio. Since these turbulence models compute the Reynolds stress based on the eddy viscosity assumption, they neglect the rotational part of the velocity gradient tensor. In the k-ϵ model, two transport equations are used to estimate the turbulent viscosity: one equation for the kinetic energy (k), and another for its dissipation (ϵ). These properties are then employed to calculate the turbulent viscosity. Both models are computationally "cheap" when compared to the RSM. The validity of the Boussinesq hypothesis is a topic of discussion within the scientific community when simulating the flow in cyclonic separators, since it has been observed that turbulence has different magnitudes depending on the direction along which it is observed – this obviously cannot be captured with the use of a scalar turbulent viscosity. The use of isotropic models to predict turbulent flow in cyclones can lead to inaccuracy in the calculation of flow, particularly in the subprediction of tangential velocity. As a result, the pressure drop in the cyclone is also under-predicted, as well as particle collection efficiency.

Algebraic Reynolds Stress Models (ARSM) are an alternative approach to constitute a non-linear stress relationship that includes the effect of the rotational part of the velocity gradient tensor. This feature assists ARSM to capture the three-dimensional turbulent features accurately (Wallin and Johansson, 2000). Hence, the present study evaluates an anisotropic model to calculate the turbulent stress tensor in two- and three-dimensional cyclone simulations. This model was presented by Meier (1998). The present implementation of the model is an improvement of an earlier study (Luciano *et al.*, 2017), which was extended to work in three-dimensional geometries. It was written as a library in an open source CFD code.

2. MATERIALS AND METHODS

The cyclone under study was designed by the research group, and is installed in a testing facility located at the Verification and Validation Laboratory (LVV) in the University of Blumenau (FURB). It is illustrated in Fig. 1a, and its dimensions are defined in Tab. 1. Experimental measurements were made by Balestrin *et al.* (2017), which used the

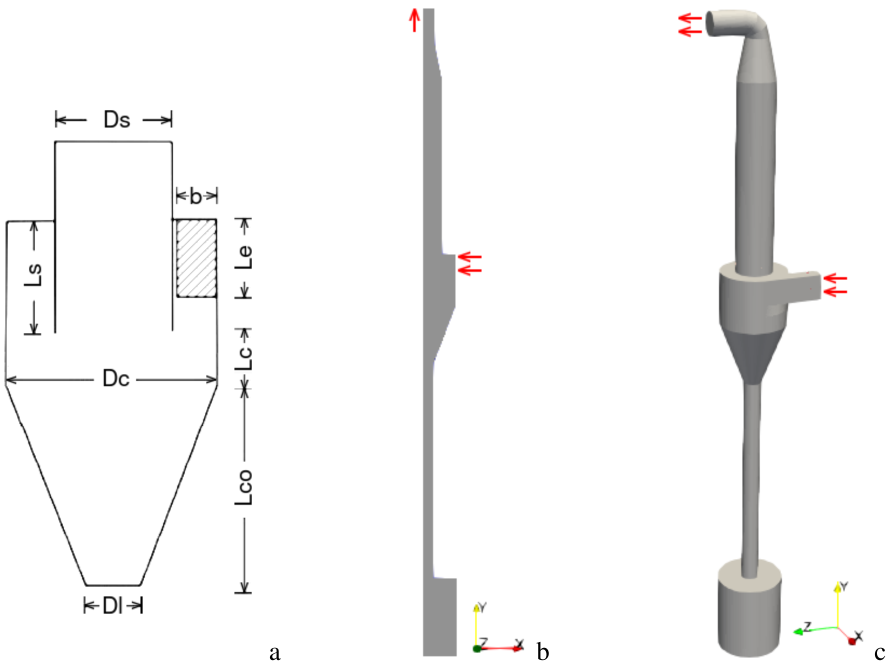

Figure 1. (a) cyclone dimensions, (b) 2-D geometry and (c) 3-D geometry.

Table 1. Cyclone dimensions.

Dimension	value (mm)
D_s	164
D_l	76
L_e	110
L_s	160
L_c	75
L_{co}	285
b	60

Stereo Particle Image Velocimetry (PIV) technique to acquire velocity profiles at the distances of 20, 40 and 60 mm from the vortex finder.

For the numerical representation of this cyclone, both two-dimension and three-dimension geometries were used. They are illustrated in Figs. 1b and 1c, respectively. Air was considered as working fluid. The inlet velocity used was 12.25 m/s, in accordance to one of the experimental essays.

A transient formulation was used to carry out the numerical simulations. A single-phase, turbulent approach was used to simulate the air flow. The continuity equation is expressed as

$$\frac{\partial}{\partial t}\rho + \nabla \cdot (\rho U) = 0 \quad , \tag{1}$$

where ρ is the density, μ the viscosity, $\mathbf{U}$ is the averaged velocity, and the momentum transport equation is

$$\frac{\partial}{\partial t}\rho U + \nabla \cdot (\rho U U) = \nabla[\mu(\nabla U + (\nabla U)^T)] + \tau^{\mathbf{Re}} - \nabla p \quad , \tag{2}$$

where p is the pressure. The $\tau^{\mathbf{Re}}$ term in Equation 2 refers to the Reynolds tensor. Here, the Reynolds tensor is preserved, but its components are recomputed from the Prandtl mixing length and k-ϵ turbulence models. In isotropic models, $\tau^{\mathbf{Re}}$ is usually expressed as

$$\tau^{\mathbf{Re}} = -2\mu^t \mathbf{D} \tag{3}$$

In the hybrid turbulence model, each component of the Reynolds tensor is computed by

$$\tau_{ij} = \left(\mu^t_{Pr} M_{ij} + \mu^t_{k\epsilon}(1 - M_{ij})\right) D_{ij} \quad , \tag{4}$$

where $\mathbf{M}$ is a logical matrix used to select one model for each component of the Reynolds tensor depending on the direction considered. It is adjusted to locate the tangential direction in each position of the mesh, since OpenFOAM uses a cartesian coordinate system. In the simulation of the cyclonic separator, all components associated with the tangential direction were obtained through the mixture length model, whereas the others through the k-ϵ model to guarantee the Galilean invariance of the Reynolds stress. As an example, when the tangential direction is aligned to the Y direction, $\mathbf{M}$ assumes the values of

$$M = \begin{vmatrix} 0 & 1 & 0 \\ 1 & 1 & 1 \\ 0 & 1 & 0 \end{vmatrix} \tag{5}$$

In Equation 4, $\mu^t_{k\epsilon}$ and μ^t_{Pr} refers to the turbulent viscosity obtained using the k-ϵ and the Prandtl mixing length models, respectively.

The transport equations for the turbulent kinetic energy (k) and its dissipation (ϵ) are expressed as

$$\frac{\partial}{\partial t}\rho k + \nabla \cdot (U k) = \nabla\left(\frac{\mu^{eff}}{\sigma_k}\nabla k\right) + G - \rho\epsilon \quad , \tag{6}$$

$$\frac{\partial}{\partial t}\rho \epsilon + \nabla \cdot (U \epsilon) = \nabla\left(\frac{\mu^{eff}}{\sigma_\epsilon}\nabla \epsilon\right) + (C_1 G - C_2\rho\epsilon)\frac{\epsilon}{k} \quad , \tag{7}$$

where G is the turbulence production, C_1=1.44, C_2=1.92, σ_k=1.0 and σ_ϵ=1.3. Using this model, the turbulent viscosity ($\mu^t_{k\epsilon}$) is calculated by

$$\mu^t_{k\epsilon} = C_\mu \rho_l \frac{\sqrt{k}}{\epsilon} \quad , \tag{8}$$

where C_μ=0.09.

Using the Prandtl mixing length model, the turbulent viscosity is calculated by (Meier and Mori, 1999; Pericleous, 1987)

$$\mu^t_{Pr} = \mu^t_{Pr,in} + (l\, r)^2\sqrt{tr(S:S)} \quad , \tag{9}$$

where r is the radial position, S is the strain rate tensor, and $\mu^t_{Pr,in}$ is the turbulent viscosity at the inlet. In this study, l=0.034, and $\mu^t_{Pr,in}$ is given by:

$$\mu^t_{Pr,in} = \rho\, \ell \sqrt{k_{in}} \tag{10}$$

where ℓ is the turbulent length scale at the inlet, considered as being 10% of the hydraulic diameter.

Tests were performed using an axisymmetric representation of a cyclone, adopting a transient formulation for the solution of the single-phase transport equations.

The meshes used to simulate both the 2D and the 3D geometries of the cyclone were composed of hexahedral volumes, built using the blockMesh software. Meshes with three sizes were evaluated, and the uncertainty due to the grid refinement was calculated using the GCI method (Roache, 1997). To evaluate the simulations in two dimensions, meshes with

3658, 5964 and 13016 hexahedral control volumes were used. To evaluate the simulations in three dimensions, meshes with 157840, 339248 and 641886 hexahedral control volumes were used.

The turbulent flow was simulated using a transient formulation with the pimpleFoam solver, available in OpenFOAM v7. To carry out the simulations, the Courant number was fixed at 0.5, and relaxation factors of 0.1 were applied to all variables. Cases were simulated for about 3 s of flow, which was observed to be enough to develop the flow profiles.

The calculations were performed in a Dell XPS station with 16 GB RAM and a CPU with 3.6 GHz.

To calculate the pressure drop, Balestrin *et al.* (2017) measured 4 points close to the walls at the inlet of the cyclone, and another 4 points above the overflow. This procedure was used to evaluate the pressure drop in the 3D simulations. For 2D simulations, only one point close to the walls at the inlet, and another above the overflow, were considered.

Finally, tangential velocity profiles were measured 20 mm below the vortex finder, in order to compare them with experimental measurements.

3. RESULTS AND DISCUSSION

The GCI method was used to evaluate the uncertainty of the results due to the mesh refinement. The property evaluated was the pressure drop: it is expected that the model used will be able to accurately predict the velocity profiles developed inside the cyclone. The failure to predict the tangential velocity leads, as a consequence, to inaccuracy in the estimation of the pressure drop.

Considering the 2D simulations, the GCI method indicated that the refined mesh provided results within 1.85% of accuracy; the intermediate mesh had an uncertainty of 8.58%. Thus, to carry out simulations in 2D, the most refined mesh was used.

For 3D simulations, the GCI method indicated that the refined mesh has an uncertainty of 0.58%, and the intermediate mesh 2.91%, considering the pressure drop. Hence, the intermediate mesh was used to conduct 3D simulations.

The vorticial flow inside the cyclone is strongly dependent on the wall condition. This condition is associated with a combination of a no-slip and a free vortex condition (Meier, 1998). There are empirical correlations according to the cyclone geometry. Since the evaluated cyclone was different from those observed in classic geometries (e.g. Lapple and Stairmand configurations), we evaluated the use of both a no-slip velocity, free-slip condition (which represents the free vortex), and a prescribed value for the tangential velocity (equal to the inlet velocity) at the walls of the cyclone, in the cylindrical body and conical region. In this third condition, both the axial and radial components of the velocity were set to 0. Figure 2 presents the comparison between simulated results obtained using these wall conditions in the 2D geometry (shown in Fig. 1b). The experimental data shown in this figure was acquired using a stereo PIV setup. The uncertainty of the experimental measurements (10%) is represented by error bars.

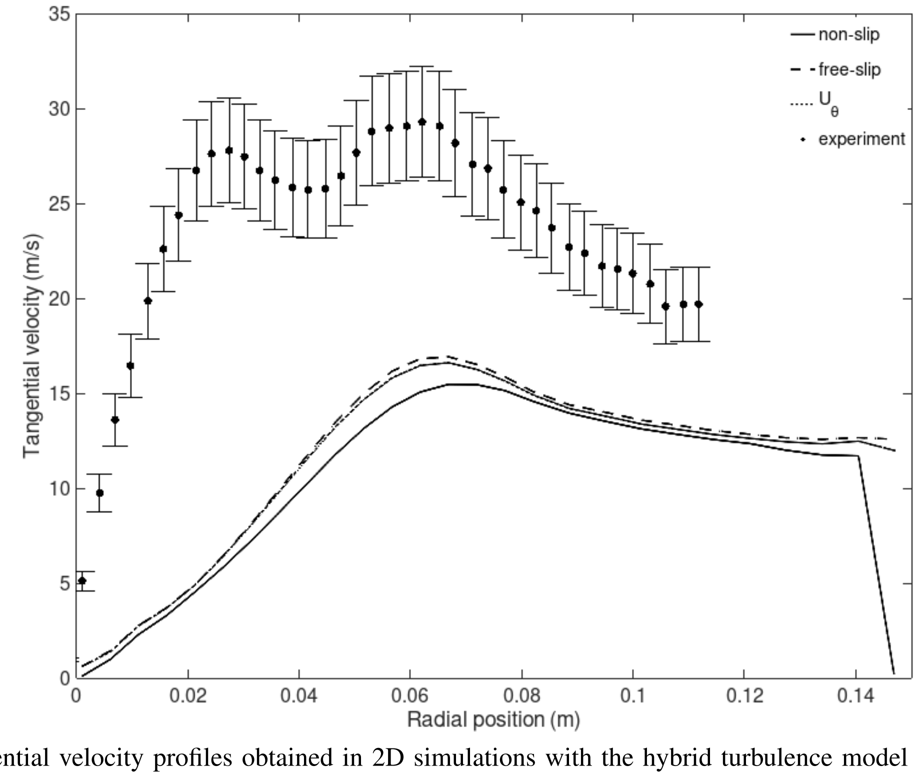

Figure 2. Tangential velocity profiles obtained in 2D simulations with the hybrid turbulence model and different wall conditions.

Figure 2 shows that using the no-slip condition, the predicted tangential velocity profile presents a sharp increase near the wall, reaching a velocity magnitude close to the inlet velocity. From the wall, the flow develops the Rankine profile into the center of the cyclone. The peak in the predicted tangential velocity is located near a radial position of 0.065 m, close to the peak position found in the experimental measurements. Using the free-slip condition, the velocity is already high at the walls. It enables the flow to develop slightly higher tangential velocities inside the cyclone. This velocity profile was very close to the one resulting from the use of a prescribed value at the walls. Still, the maximum values for the tangential velocity differ: 15.48 m/s (non-slip), 16.93 m/s (free-slip) and 16.62 m/s (prescribed velocity), against the experimental value of 29.31 m/s. Balestrin *et al.* (2017) highlighted that the absence of the complete outlet geometry for this cyclone may result in a subprediction for the tangential velocity, due to the constraint after the vortex finder.

Since the implementation of the hybrid turbulence model was made with the intention to use it in three-dimensional geometries, it was necessary to verify the suitability of the written code to predict the tangential velocity profile in different orientations, before applying it in the simulation of a 3D case. Thus, simulations were carried out using 2D geometries aligned with different orientations: the geometry seen in Fig. 1b), which was aligned with the XY plane, was then rotated 45° and 90° around the Y axis: at 0° rotation, the tangential velocity is aligned to the Z direction; at 90° rotation, the tangential velocity is then aligned to the X direction; and, 45° rotation, the tangential velocity is not perfectly aligned to any of the cartesian directions. See axis at the bottom of Figure 1b.

Figure 3 shows the results obtained with geometries in three different orientations. The results obtained for the geometry rotated 0° and 90° were exactly the same. When the geometry was rotated 45°, a small deviation was observed when compared to the other profiles. Nonetheless, the obtained results indicates that the implemented model is suitable to predict the flow behavior at different orientations.

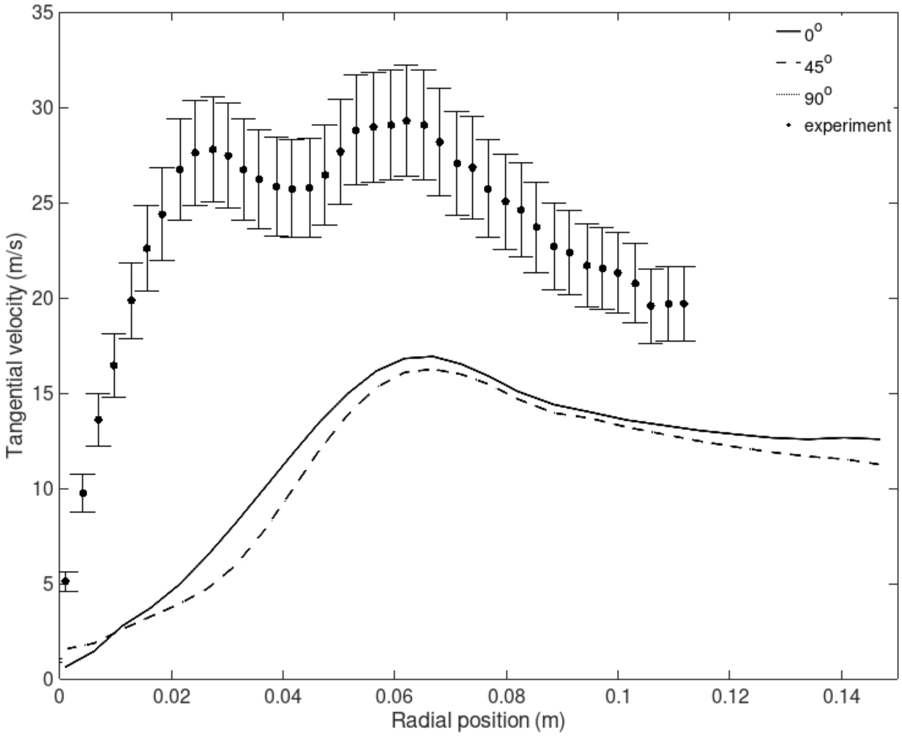

Figure 3. Tangential velocity profiles obtained in 2D simulations with the geometry oriented at different angles.

After these preliminary tests, simulations were carried out using the 3D geometry presented in Fig. 1c, considering again different wall conditions for the velocity, in order to evaluate the resulting velocity profiles. The obtained results are shown in Fig. 4. Comparing the tangential velocity profiles obtained with the 3D geometry against the profile obtained in a 2D geometry (Fig. 2), it can be seen that the use of the complete domain promotes higher tangential velocities, but it is still considerably lower than the experimental values (Fig. 4) when considering the no-slip condition at the cyclone walls.

Considering a free-vortex condition at the cyclone walls, it can be noticed that the resulting tangential velocity profile overpredicted the experimental values. In spite of the closer value for the maximum tangential velocity, the free-vortex condition did not predicted the decrease in the tangential velocity in the region close to the walls, which is present in the experimental data.

The use of a prescribed tangential velocity at the cyclone walls seemed to result in the best comparison against the experimental data, according to the profiles presented in Fig. 4. In this condition, the velocity starts from 12.25 m/s at the walls, increasing to a maximum tangential velocity within measurement uncertainty, both in its value and position, to the

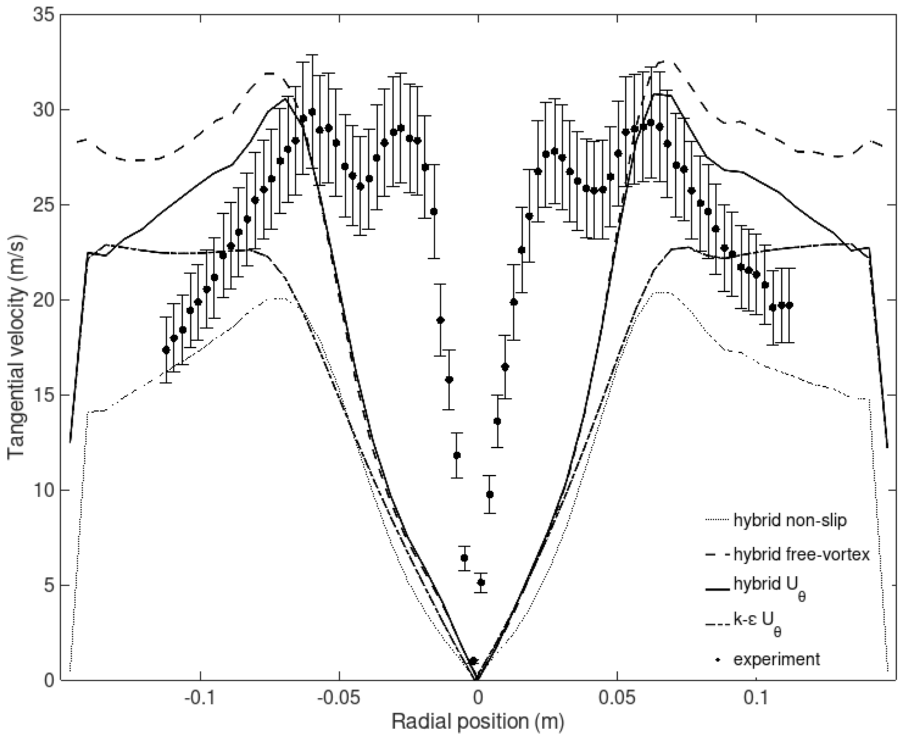

Figure 4. Tangential velocity profiles obtained in 3D simulations.

experimental value.

Similar to the non-slip condition, using the hybrid turbulence model and a prescribed tangential velocity at the walls resulted in velocity profiles with a similar shape when compared to the experimental values. This similarity was not achieved by using the standard k-ϵ turbulence model. Even considering a prescribed tangential velocity at the walls, this model failed to predict the experimental tangential velocity profile: instead of the expected Rankine profile, this model produce a smooth curve without a clear position for the velocity peak.

Comparing all results, Figure 4 shows that the use of the hybrid model in the 3D geometry, coupled with a prescribed tangential velocity at the walls, provided the best prediction of the tangential velocity profile, both qualitative and quantitative. In addition, considering the obtained results, all combinations using the hybrid turbulence model provide better qualitative results than the standard k-ϵ model. It should be noted that this model has computational cost similar to a two-equations turbulence model, that is lower than the Reynolds Stress Model, which commonly used in cyclone simulations. The RSM model was used by Balestrin *et al.* (2017) in a 3D mesh containing $\approx$4 million elements, which enabled to predict the secondary tangential velocity peak observed in the experiments (which was attributed to the outlet configuration). Considering the present study, the mesh refinement could be crucial for the accurate prediction of the velocity profiles. For the purpose of the implementation and verification of the hybrid model, the obtained results were considered satisfactory.

Figure 4 shows that, unlike the 2D cases, the predicted results are very sensible to the wall conditions in 3D simulations. In this sense, the use of higher degree of refinement near the wall could also improve the prediction of the velocity in this region. However, it should be noted that the k-ϵ model provides a poor prediction of the flow when using too refined meshes. It should be also noted that the hybrid model is somewhat unstable, requiring the use of relaxation factors in order to stabilize the solution.

4. CONCLUSIONS

A hybrid turbulence model, reported in the literature, was implemented in a CFD code to be able to simulate 3D geometries. The produced code was verified by considering a series of 2D geometries with different orientations. In the evaluations made in the present study, the hybrid model predicted values closer to the experimental ones for the tangential velocity when compared against those obtained using the standard k-ϵ model, at similar computational cost.

The use of a prescribed tangential velocity at the cyclone walls enhanced the obtained velocity profiles. Moreover, the use of the hybrid turbulence model in a three-dimensional representation of the cyclone resulted in the best prediction of the rotating flow. This demonstrates the applicability of the model evaluated in this study to simulate the flow in complex geometries.

5. REFERENCES

Balestrin, E., Decker, R.K., Noriler, D., Bastos, J.C.S.C. and Meier, H.F., 2017. "An alternative for the collection of small particles in cyclones: Experimental analysis and cfd modeling". *Separation and Purification Technology*, Vol. 184, pp. 54–65. doi:10.1016/j.seppur.2017.04.023.

Luciano, R.D., Rosa, L.M., Balestrin, E., Noriler, D. and Meier, H.F., 2017. "Implementação de modelo anisotrópico de turbulência para simulação do escoamento giratório em ciclone". *RETEC. Revista de Tecnologias*, Vol. 10, pp. 62–74.

Meier, H.F., 1998. *Modelagem fenomenológica e simulação bidimensional de ciclones por técnicas da fluidodinâmica computacional*. Ph.D. thesis, UNICAMP, Campinas.

Meier, H.F. and Mori, M., 1999. "Anisotropic behavior of the reynolds stress in gas and gas-solid flows in cyclones". *Powder Technology*, Vol. 101, pp. 108–119. doi:10.1016/S0032-5910(98)00162-4.

Pericleous, K.A., 1987. "Mathematical simulation of hydrocyclones". *Applied Mathematical Modelling*, Vol. 11, pp. 242–255. doi:10.1016/0307-904X(87)90139-9.

Roache, P.J., 1997. "Quantification of uncertainty in computational fluid dynamics". *Annual review of fluid Mechanics*, Vol. 29, No. 1, pp. 123–160. doi:10.1146/annurev.fluid.29.1.123.

Wallin, S. and Johansson, A.V., 2000. "An explicit algebraic reynolds stress model for incompressible and compressible turbulent flows". *Journal of fluid mechanics*, Vol. 403, pp. 89–132. doi:10.1017/S0022112099007004.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.