Influence of Turbulence Modeling on Velocity Profiles for Cyclone Separators

Influence of Turbulence Modeling on Velocity Profiles for Cyclone Separators

Gianluigi Santillo, Andrea Rapisarda, Alessio Desando, Elena Campagnoli

Politecnico di Torino, Dipartimento Energia

Torino, Italia

Email: **gian.santillo@gmail.com; andrea.rapisarda@polito.it; alessio.desando@polito.it; elena.campagnoli@polito.it;**

Summary

Newer aircraft engines are designed in order to obtain both the best performances and the lowest environmental impact, reducing the amount of polluting elements emitted in the atmosphere. The improvement of engine lubricating circuits is related to this task, aiming to a more efficient lubricant recycling when flowing into filtering devices. Cyclonic separators constitute one stage of the filtering phase. They are simple devices whose working principle consists in using centrifugal forces to separate two phases from one another, e.g. solid particles from a fluid or two fluids having different densities. The present work focuses on the impact of the turbulence models in the simulations of cyclone separators. The objective is to provide guidelines for future numerical analyses, paying special attention to the correct simulation of velocity profiles, which play a very important role in the particle separation process.

Keywords

Gas cyclone, CFD, Turbulence models, Velocities profiles, ANSYS

Introduction

Cyclone separators are devices employed in many industrial processes. Since there are no moving (then no wearable) parts, their simple geometry allows an easy manufacture. They have become interesting for some aerospace application because it has been proven that the separation process is affected by centrifugal forces, inertia and drag forces much more than by gravitational ones. This means cyclones can be positioned indifferently in a horizontal or vertical way, encouraging applications involving a good space optimization.

The flow enters the tangential inlet, descending with an axial spiral motion, while centrifugal forces separate the air flow from the particles, which are collected by means of an underflow pipe. At a certain height – location indicated as the vortex end position – the axial velocity inverts its direction making the flow ascend. This peculiar phenomenon is apparently originated by the pressure and flow field inside the cyclone, and not directly by its geometrical features. So a double vortex structure is formed, with an inner vortex that leads the flow out through a central duct called vortex finder. This central duct extends toward the inside of the cylindrical chamber to both shield the inner vortex from the outer one and to stabilize it [1].

So the principle which cyclones are based on is the creation of two vortexes, an outer one directed downwards and carrying the air/oil bi-component fluid, and an inner one pointing upwards with a limited dispersion of oil particles in the flow (Figure 1) [2].

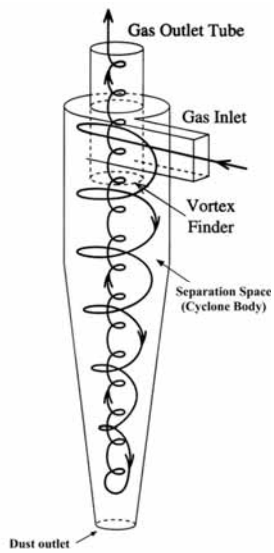

Figure 1: Vortex flow in cyclone separators [3]

The separation takes place because of the difference in density between the two phases composing the mix injected inside the cyclone. The particles, with a higher density, are pushed against the walls by a strong centrifugal force. Colliding they lose momentum and begin to slide along the walls towards the lower part of the conical section (the cone tip), being then picked up in a dust collector.

Speaking about experimental investigations, numerous examples can be found in literature. To obtain the flow field pattern the majority of these studies used either laser Doppler velocimetry (LDV) or particle image velocimetry (PIV), i.e. the two procedures that showed to be the most precise and reliable. These techniques have been employed to measure pressure drop and collection efficiency for the Stairmand high-efficiency cyclone at different flow rates, or the mean and fluctuating velocity components for gas cyclones (as A. J. Hoekstra [4] did). Some studies were also focused on the effect of cyclone geometry on separation efficiency. Although experimental tests are a necessary step to achieve always newer and more efficient solutions, the implementation of numerical models during the preliminary design phase could be a valid alternative to expensive experimental tests. The processing power of modern computers allows indeed the simulation of more complex and detailed phenomena than the past years.

The present work concerns the implementation of a numerical model for cyclone separators, focusing on different turbulence models. The primary difficulties in studying cyclone separators flow field arise from their highly anisotropic turbulence. This makes most of the first order turbulence approximations, like the popular $k - \varepsilon$ model, not reliable to predict the flow characteristics. Several attempts were made to overcome this limitation. Boysan et al. [5] were among the first researchers to report CFD studies of cyclone flows, and their early studies (their test were performed in 1982) already showed that the standard $k - \varepsilon$ turbulence model is not able to accurately simulate this kind of flow and that at least a second order turbulence closure – e g. Reynolds Stress Model – was needed to capture the anisotropy and achieve realistic simulations of the flow inside a cyclone. Gronald et al. [6] performed an experimental LDV analysis on the Obermair cyclone. Then, a numerical approach was set up considering two different turbulence models to demonstrate which one is more accurate or expensive in terms of computational costs. Many studies have been performed since then, and all of them made clear that the choice of the turbulence model is the most critical aspect of CFD simulations of cyclone separators. While it is possible to feel the urge of using a higher order turbulence model, it is also important noticing that as larger ranges of time and length scales are being resolved, the computational requirements increase awfully. Then, a trade-off between accuracy and speed of computation is needed.

Numerical Model

There are a large number of cyclone designs, depending on the application required. For the present work, the Obermair geometry [6] was chosen. The numerical investigations are focused on the impact of turbulence models in simulating the flow field inside cyclone separators. The objective is to provide guidelines for future numerical analyses, paying special attention to the correct simulation of velocity profiles, which play a very important role in the particle separation process.

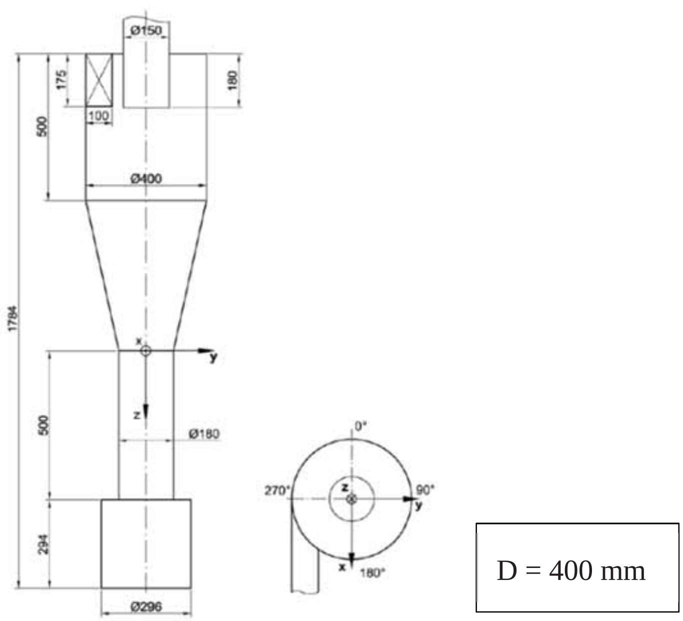

Figure 2: Obermair geometry [6].

The numerical models have been implemented by using the CFD commercial software ANSYS CFX 14.5. The accuracy of numerical results depends on many parameters, such as proper mesh settings and the discretization of the advection terms. The use of hexahedral grids for the main flow region and a second order accurate advection scheme have shown significant improvements in CFD predictions, mainly for those peculiar unsteady structures that characterize the flow in a cyclone. However, this choice depends on the turbulence model adopted for the simulation: while for RANS (Reynolds Averaged Navier-Stokes) models a domain discretization made with about 5,280,000 nodes hexahedral elements provided the best results, the LES (Large Eddy Simulation) model required a tetrahedral grid, with a similar number of nodes. This choice allows obtaining a more accurate solution since the grid is isotropic and undue effects which may arise with anisotropic grids (i.e. hexahedrons) are not present [7].

In steady state simulations, the variables do not show strong time dependence and the system is assumed to be in conditions reached after a relatively long time interval. They therefore require no real time information to describe them [7, 8].

This approach is less accurate to predict the features of the flow field in great details, like the vortices flow, but can be employed to provide simulations with initial conditions. However, for the steady state simulation it is necessary to define the calculation domain first, i.e. the space region where equations are solved (thermo fluid), and then the boundary conditions on the surfaces enclosing the fluid domain. The following table shows the boundary conditions set for the analysis.

Fluid	Air
Temperature	20° C, Isothermal
Inlet velocity	12.68 m/s
Vortex finder	Zero Gradient
Wall	No slip condition
Advection Scheme	High Resolution
Turbulence Numerics	High Resolution
Max. Iterations	2000
Timescale Control	Physical Timescale
Timescale	$5 \cdot 10^{-4}$
Residual Type	RMS
Residual Target	10^{-4}

Table 1: Boundary conditions

The outlet boundary has been set with the zero gradient condition, expressed as:

$$\frac{\partial}{\partial n}(u, k, \varepsilon) = 0 \tag{1}$$

In other words, the diffusion flux for the entire variables in exit direction is assigned as zero [9].
Transient (or unsteady) simulations require real time information to determine the time intervals at which the solver calculates the flow field. In order to avoid convergence issues, a transient should be initialized as a small perturbation of a converged steady state result; this also may reduce the time of the simulation. This approach is more accurate to predict the features of the flow field in great details on cyclone separator, but requires more setup information, as showed here below.
For a time-dependent problem, time-step is the numerical discrete interval on which the solution integration is performed. However, there is not a universal stability criterion helping to determine the time-step, but to properly model transient phenomena it is important using a time-step of at least one order of magnitude smaller than the smallest time constant in the system (residence time for cyclone separator) [3]. The average residence time is estimated by the follow equation:

$$\tau = \frac{V}{q} = \frac{\text{cyclone volume}}{\text{gas volume flow rate}} \tag{2}$$

Another important verification to ensure the convergence of the simulation is the Courant number, defined as:

$$C = \frac{u \cdot \Delta t}{\Delta x} \tag{3}$$

where u is the airflow speed, Δt the time-step and Δx the mesh size [8].
In transient simulations, a good value for the Courant number is $2 < C < 10$. Knowing the grid size and the inlet velocity, it is possible to easily get a first evaluation of the time-step from the equation.

The time-step has been set equal to $5 \cdot 10^{-4} s$ (except for the LES turbulence modeling, which has further constraints).
There is a great number of turbulence models which have been implemented on commercial software products for CFD analyses, and they found a large employment in many industrial fields. The aim of this work is to identify the approaches, which are a good trade between an adequate modeling of the physical phenomena and the computational costs. The most common turbulence models are the RANS (Reynolds-Averaged Navier-Stokes) and the LES (Large Eddy Simulations).
The first approach consists in the closure of the Navier-Stokes equations, using the Reynolds decomposition, which consists in decomposing an instantaneous quantity into its time-averaged and fluctuating quantities. The several RANS employed in the presents work are the following:

- The 'standard' $k-\varepsilon$ model is a classical two-equation model.
- The $k-\omega$ model, also models the turbulence with two equations. This model is appropriate to obtain better boundary layer modeling and flow separation prediction.
- The RSM (Reynolds Stress Model) is a two-equation method that provides best results for rotating flow simulations.

The LES modeling is quite different from the RANS methods, since the equations are filtered from the smaller time and length scales. To perform this approach, very fine meshes and time steps are required, so the computational costs highly increase. The LES simulation could be a valid alternative to RANS where high Reynolds numbers occur and the flow is unsteady with coherent structures, such as in cyclones [9].

Numerical Results

Steady-state Results

The numerical results are shown in terms of axial and tangential velocity profiles at the heights indicated in Figure 3. Each graph compares the trends obtained for the several turbulence models to the experimental data [6].
Due to the unsteady nature of the two-vortex flow arising in the cyclone separator, the steady-state simulations provide poor results, irrespective of the turbulence model employed, as shown in Figure 3. The velocity profiles plots for the other heights have not been reported since they all present a similar behavior and the physical phenomenon is not adequately simulated.

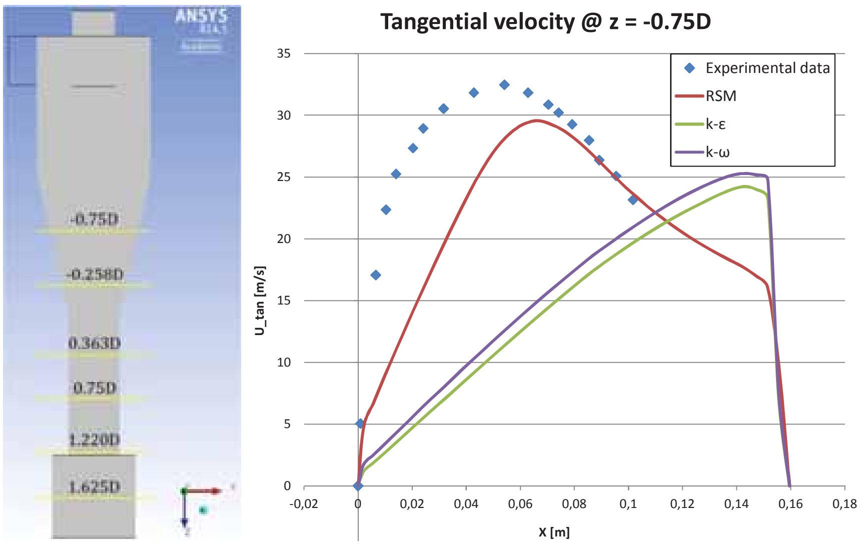

Figure 3: Heights where the velocity profiles are analyzed (left); tangential velocity profiles, steady conditions (right).

Transient Results

In order to obtain more accurate and reliable results, it is necessary to perform transient analyses, which are highly demanding in terms of computational time. Besides the meshing and time-step settings, which have been already discussed in the numerical modeling paragraph, a proper initialization is also important to ensure convergence. All the transient simulations have been indeed initialized with a steady-state simulation performed with the same turbulence model, except for the LES, whose initialization was made with the RSM steady-state and transient results. The number of iterations is 3500, in order to simulate a time interval wide enough to allow the flow forming the two-vortex structure. For the LES the number of iterations is 20000, since the time-step is smaller ($10^{-4}s$), plus 1500 iterations of the RSM unsteady initialization.
The velocity profiles obtained with the unsteady simulations are shown at several heights for the axial and tangential components, comparing the results provided by the turbulence models with the experimental data [6].

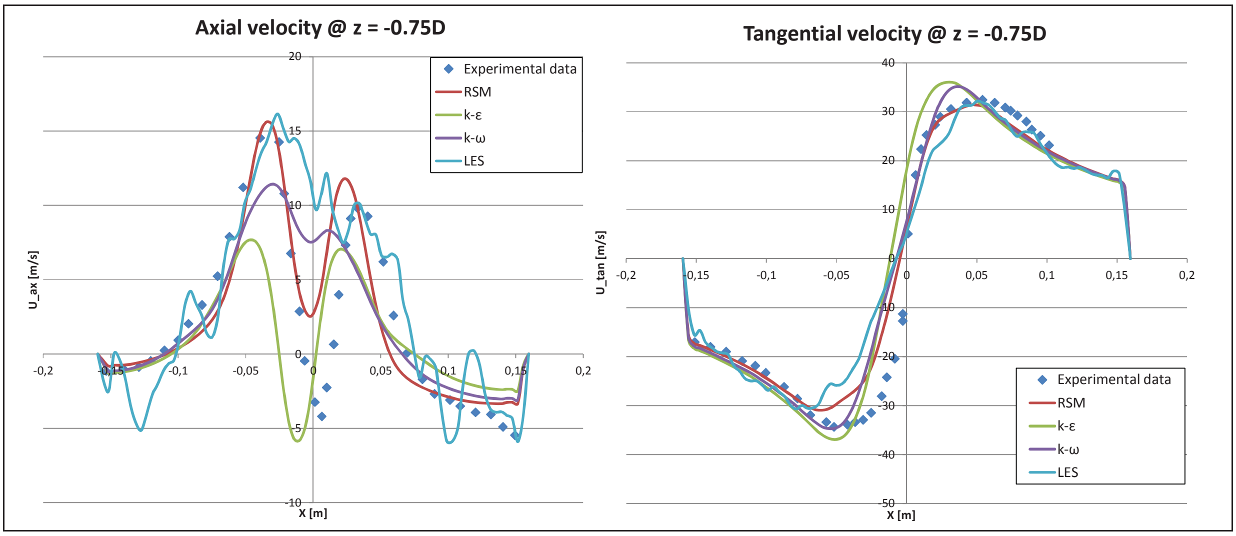

Figure 4: Unsteady velocity profiles, $z = -0.75D$.

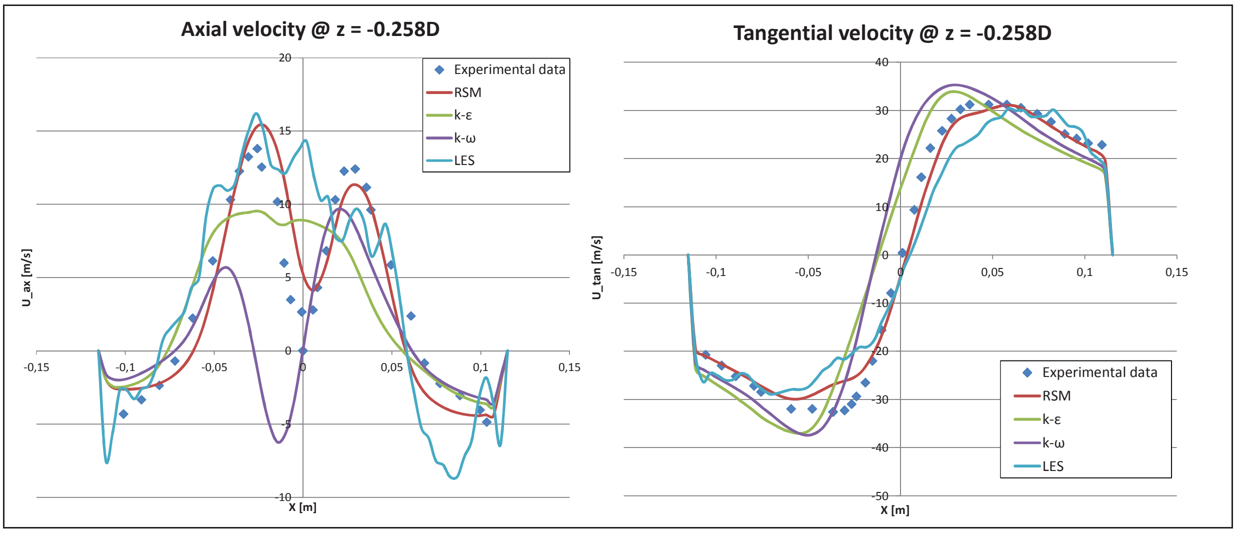

Figure 5: Unsteady velocity profiles, $z = -0.258D$.

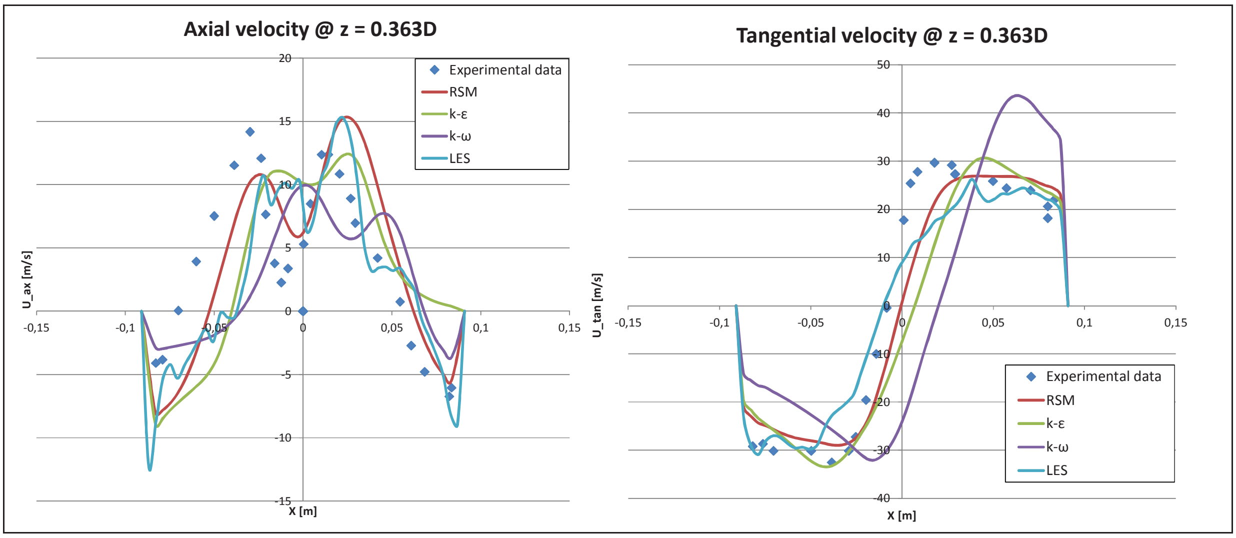

Figure 6: Unsteady velocity profiles, $z = 0.363D$.

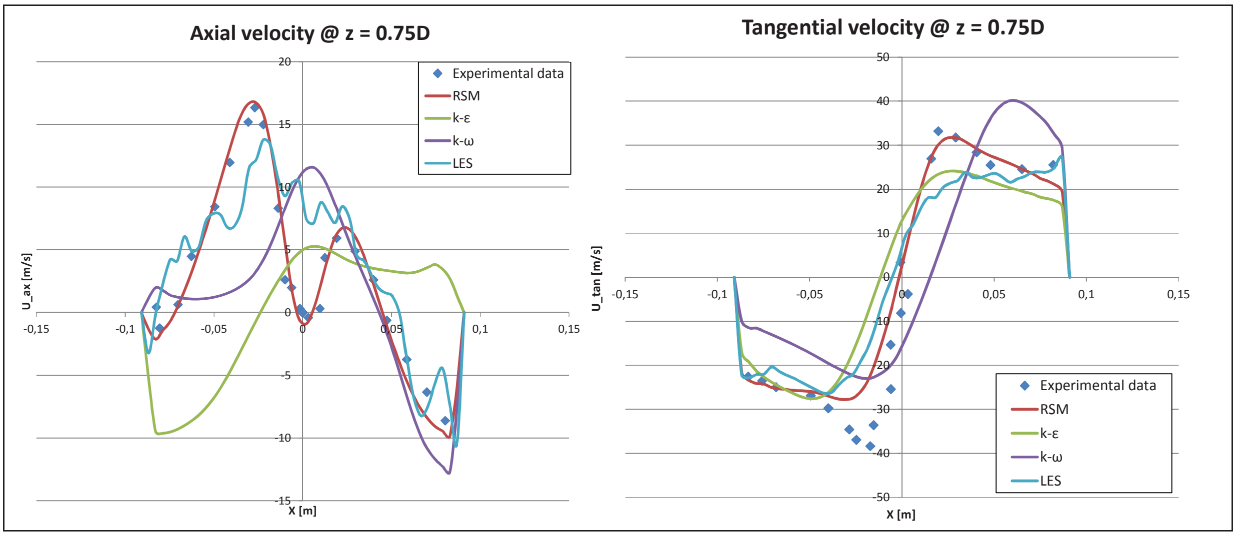

Figure 7: Unsteady velocity profiles, $z = 0.75D$.

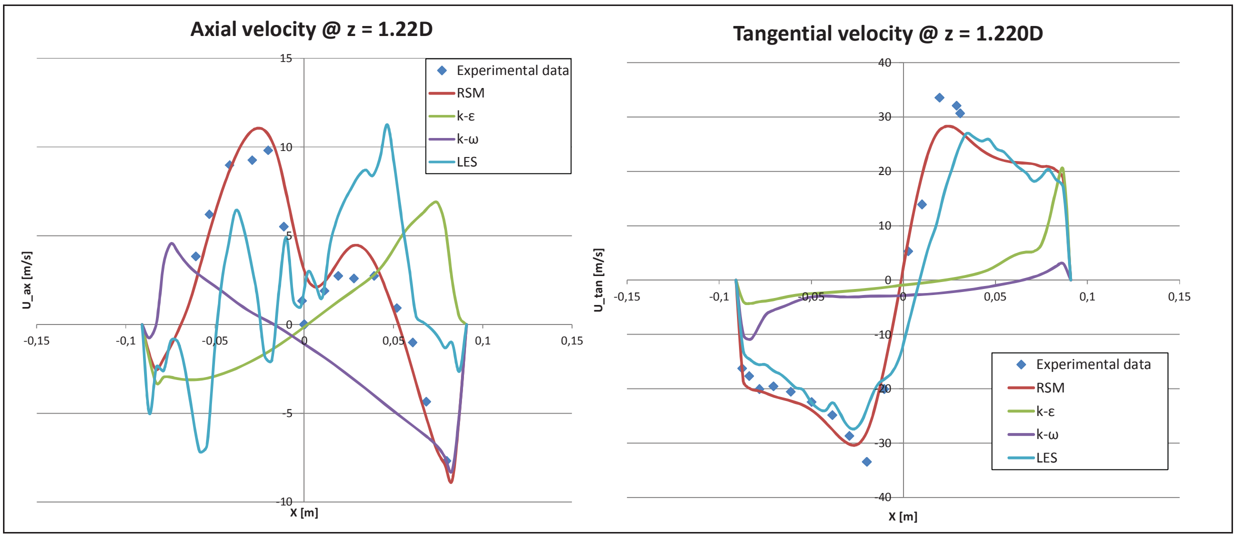

Figure 8: Unsteady velocity profiles, $z = 1.22D$.

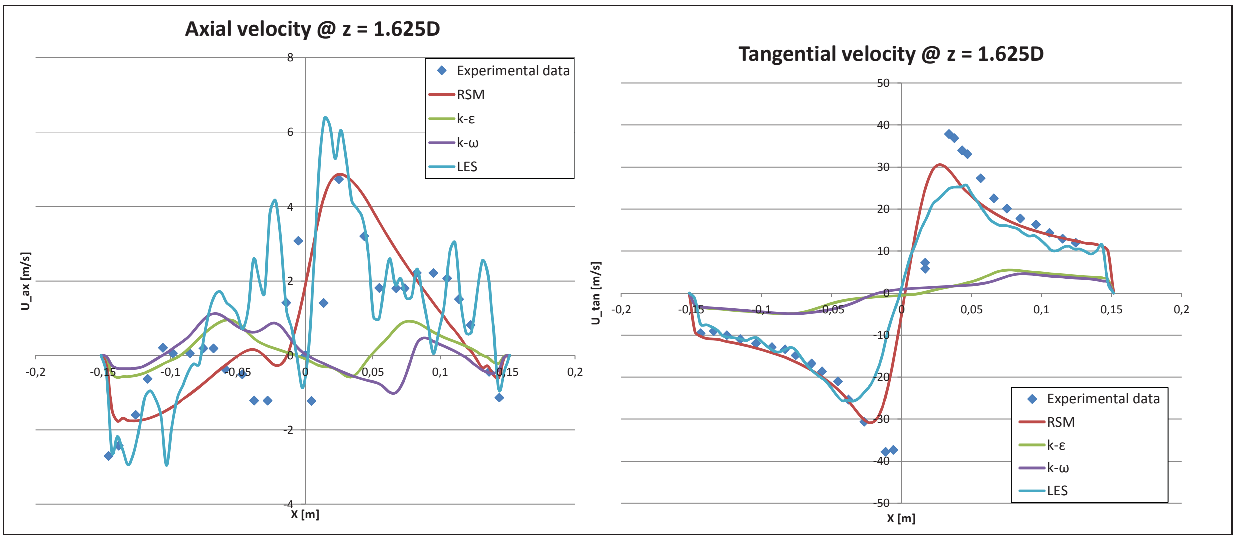

Figure 9: Unsteady velocity profiles, $z = 1.625D$.

In order to obtain a further comparison between the curves, the mean percentage error[1] is shown in Tables 2 and 3. The error has been defined as:

$$e = \frac{\left(\int_x u\right)_{CFD} - \left(\int_x u\right)_{EXP}}{\left(\int_x u\right)_{EXP}} \cdot 100 \qquad (4)$$

z	k − ε	k − ω	RSM	LES
$-0.75D$	-42	-17	-10	+23
$-0.258D$	-30	-50	-15	+10
$0.363D$	-5[1]	-35	+4[1]	-11
$0.75D$	-18	-6[1]	+10	+17
$1.22D$	-43	-37	+1	-17
$1.625D$	-73	-68	+7	+33

Table 2 : Axial velocities : relative errors at different heights

z	k − ε	k − ω	RSM	LES
$-0.75D$	-5	-10	-13	-18
$-0.258D$	-1	+5	-7	-12
$0.363D$	-7	-1[1]	-7	-18
$0.75D$	-24	-17	-9	-22
$1.22D$	-86	-87	-7	-19
$1.625D$	-84	-85	-14	-27

Table 3 : Tangential velocities : relative errors at different heights

[1] Calculated errors are very low even though the numerical trends do not accurately fit the experimental results. This is due to the definition given for the error.

The differences among the several turbulence models are underlined by the results obtained for the axial velocity profiles, showed in Figures from 4 to 9 on the left. In the first place, the $k-\varepsilon$ model cannot replicate the M-shape due to the two-vortex structure. This result can be observed at all the heights analyzed, except for the first one, where the turbulence model follows the trend of the experimental data but underestimating the axial velocity. Neither the $k-\omega$ model is capable to predict the flow field, since the axial velocities are not in agreement with literature data at any height. The obtained numerical results are more fit instead when the third turbulence model was considered, the RSM. This model provides indeed good results for rotating flows. The M-shape for axial velocity is replicated with adequate accuracy at all the analyzed heights, even though with some underestimations. In particular, at $z = 0.75D$, the fit is very accurate, while at $z = 1.625D$ (dust-collector) is the worst one. The results provided by a LES approach with the same grid refinement used for RANS simulations are not accurate, since the M-shape is not replicated and many oscillations are present. Besides the trends showed in figures, the averaged relative error has been calculated at the several heights analyzed. Once again, the RSM show the lowest error, when compared to other approaches.

The experimental data provided for the tangential velocities show a typical S-shape [6]. For the first four heights, all the turbulence models replicate this trend. However, the RMS follows the curves shapes the best, while the others overestimate the velocity values. At the last two heights (which are respectively the end of the dipleg and the dust collector) the $k-\varepsilon$ and the $k-\omega$ totally miss the fit with the experimental trends, while the RSM and the LES approaches are still in good agreement with literature data. Table 3 shows the relative errors calculated for the several turbulence models on tangential velocities.

Conclusions

Due to the strong unsteadiness of the physical phenomenon and the complex flow structure, CFD simulations for cyclone separators are highly demanding in terms of both time and computational capability. The present work concerned their flow simulation, particularly focusing on the impact of turbulence modeling on velocity profiles. An accurate flow modeling enhances indeed the prediction of the separation efficiency. The obtained results have been compared to the literature data [6] and the turbulence has been modeled with several methods, i.e. the RANS ($k-\varepsilon$, $k-\omega$, RSM) and the LES (Smagorinsky-Lilly model) approaches. The axial and tangential velocity profiles have been analyzed at several heights in an Obermair geometry [6].

Considering a similar number of grid nodes, the LES approach required a smaller time-step to ensure convergence, thus higher computational cost and time. The three RANS models have about the same requirements in terms of simulation time. The axial velocity profiles obtained with these methods were not accurate. Among the RANS models utilized, the RSM provided the best fit with the experimental data, since it was capable to replicate the M-shape typical of axial velocity profiles. In terms of tangential velocities, characterized by a S-shape, all the four models showed a good fit with literature data, except in the region near the dust collector, where the $k-\varepsilon$ and the $k-\omega$ models failed completely.

Overall, among the analyzed turbulence models, the RSM guarantees the best choice for what concerns velocity profiles modeling, since it shows a good agreement with LDV measurements, plus its computational cost is lower than the LES approach and similar to the one required by the other RANS models.

References

[1] Alex C. Hoffmann, L. E. Stein.: "Gas Cyclones and Swirl Tubes", Springer, 2002

[2] Cristòbal Cortés, Antonia Gil.: "Modeling the gas and particle flow inside cyclone separators", Progress in Energy and Consumption Science 33, 2007.

[3] K. Elsayed.: “Analysis and optimization of cyclone separators geometry using RANS”, PhD thesis, 2011.

[4] A. J. Hoekstra., “Gas flow field and collection efficiency of cyclone separators”, Dissertation, 2000.

[5] F. Boysan, W.H. Ayer, J. A. Swithenbank, “Fundamental mathematical-modelling approach to cyclone design”, Transaction of Institute Chemical Engineers 60, (1982).

[6] G. Gronald, J. J. Derksen.: " Simulating turbulent swirling flow in a gas cyclone: A comparison of various modeling approaches", Powder Technology 205, 2011.

[7]"ANSYS CFX-Solver Modeling Guide", ANSYS® Inc. 2013

[8]"ANSYS CFX-Solver Theory Guide", ANSYS® Inc. 2013.

[9] B. Zhao, Y. Su, J. Zhang: “Simulation of gas flow pattern and separation efficiency in cyclone with conventional single and spiral double inlet configuration”, Chemical Engineering Research and Design 84, (2006).