Chemical Engineering Research and Design 136 (2018) 207–218

Contents lists available at ScienceDirect

Chemical Engineering Research and Design

journal homepage: www.elsevier.com/locate/cherd

The interaction of confined swirling flow with a conical bluff body: Numerical simulation

*Nabil Kharoua**, *Lyes Khezzar*, *Mohamed Alshehhi*

Mechanical Engineering Department, Khalifa University of Science and Technology, Petroleum Institute, Abu Dhabi, United Arab Emirates

ARTICLE INFO

Article history:
Received 4 January 2018
Received in revised form 17 April 2018
Accepted 18 April 2018

Keywords:
Swirling flow
Bluff body
Cone
Large Eddy Simulation

ABSTRACT

Large Eddy Simulations LES were conducted to study single-phase flow in a geometry with a bluff body inserted into a long pipe as a simplified case of a more complex separator. Two configurations, without bluff body and without swirl generator, were considered for comparison. While studies on swirling flows inside pipes do exist in the literature and plenty of information is available, the present simulations address the case where the swirling flow interacts with a bluff body which was scarcely considered hitherto.

The results showed a persisting core flow reversal till the bluff body location under the flow conditions considered. The swirling flow has a Rankine-vortex structure with more turbulence at its core. The bluff body undergoes the effects of recirculation zones at its front and wake regions which affect the corresponding drag and lift forces considerably. The swirl number decays from 1.5 to unity close to the bluff body. No dominant frequency was noticed in the core region. All these findings represent a starting point for an optimization work on the appropriate location and shape of the bluff body for the real separator.

1. Introduction

Swirling flows are generally used in the industry for separation, mixing and flame stabilization. They are also used in pneumatic conveying to minimize pressure drop and prevent particle deposition in pipes (Hamdani et al., 2016). The most common characteristic of swirling flows is the combination of axial and tangential velocities (Kuroda and Ogawa, 1986). Swirling flows can be confined (e.g., pipe flows) or unconfined (e.g., jets). Swirling pipe flows are usually classified based on the profile of the tangential velocity component (Kitoh, 1991; Steenbergen, 1995; Moene, 2003) yielding four types: forced vortex (solid body rotation), free vortex, combined forced/free vortex (Rankine vortex) and wall jet. The initial velocity profile is a key parameter in swirling flows. It is strongly affected by the technique used to generate the flow usually using guiding vanes or tangential inlets. These swirl generators yield either symmetric or asymmetric flows depending on their shape.

The most common parameter used to quantify the swirl intensity is the swirl number S defined by the ratio of the flux of the angular momentum to the flux of the axial momentum. Based on their experimental findings, (Kitoh, 1991; Steenbergen, 1995) have found that the swirl decay in a pipe can be represented for a certain range of swirl number by an exponential decay function. The swirl number is defined, for axisymmetric flows, as:

$$S = \frac{2\rho\pi \int_0^R r^2 uw dr}{\rho\pi R^3 u_m^2} \tag{1}$$

There exist several ways of how the swirl number is defined, Eq. (1) is based on the tangential wall shear stress normalized by the quantity $\rho\pi R^3 u_m^2$ and is adopted in the present work.

---

* *Corresponding author.*
E-mail addresses: nkharoua@pi.ac.ae (N. Kharoua), lkhezzar@pi.ac.ae (L. Khezzar), malshehhi@pi.ac.ae (M. Alshehhi).

https://doi.org/10.1016/j.cherd.2018.04.034

Nomenclature

D	Pipe diameter
k	Turbulent kinetic energy
k_{sgs}	Sub-grid-scale turbulent kinetic energy
p	Pressure
r	Local radius
R	Pipe radius
$\bar{S}_{ij}$	Filtered strain rate tensor
S	Swirl number
t	Time
$\bar{u}_i$	Filtered x_i-velocity component
u', v', w'	Fluctuating velocity components
u_m	Mean axial velocity
x_i	Cartesian coordinate
y+	Normalized wall distance
Greek letters	
ε	Turbulent dissipation rate
μ	Fluid dynamic viscosity
μ_t	Turbulent dynamic viscosity
ρ	Density
σ_{ij}	Laminar stress tensor
τ_{ij}	Subgrid-scale stress tensor

It is noteworthy to mention that the minimum axial velocity inside the pipe might be negative corresponding to a reversed flow at the core region. The literature mentions values of the swirl number greater than 0.5–0.6 for the reversal to occur (Steenbergen, 1995; Lucca-Negro and O'Doherty, 2001). The low velocity core is characterized by a low pressure which explains the deceleration of the flow at the core region due to the axial adverse pressure gradient.

Another important characteristic of swirling flows is the precessing vortex core PVC (Gupta and Kumar, 2007; Hreiz et al., 2011). Its interaction with the reversal flow generates complex structures such as kidney shape and helical bubble.

The abovementioned literature dealt with confined swirling flows in pipes without obstacles along the axial flow direction. More relevant to the present work are contributions having considered obstacles which affect the swirling flow structure noticeably. The existing literature focused on spherical obstacles mainly to study the interaction of solid particles with a turbulent fluid flow containing eddies of different sizes (e.g., Mattner et al., 2003; Atvars et al., 2009; Grinis and Tzadka, 2011). These contributions found that the swirl generates a recirculation zone upstream of the obstacle with different structures depending on the swirl number shifting the stagnation point away from the front side of the sphere. In addition, the separation point, on the lateral surfaces of the obstacle and the resulting wake vanish gradually by increasing the swirl number.

The remarkable progress of CFD techniques and computational resources allowed researchers to tackle swirling flows with their complex features such as streamline curvature and non-equilibrium conditions. The choice of the turbulence model is crucial for an accurate prediction of the velocity profiles in swirling flows. The work of Kobayashi and Yoda (1987) represents one of the first unsuccessful attempts to predict a swirling flow using the k-ε turbulence model. The k-ε turbulence model and similar models, based on the assumption of turbulence isotropy, do not perform correctly for the complex swirling flow. The weakness is overcome using the Reynolds Stress Model RSM which calculates the stresses directly. Kitoh (1991) compared his experimental results with those predicted by the RSM model. He found that the RSM model with the LLR (Launder, 1989) pressure-strain term under-predicts the streamwise Reynolds stress component by 20%. He concluded that the model constants needed to be adjusted to improve the accuracy of the CFD results. More recent contributions, in the field of hydrocyclones (e.g., Brennan, 2006), confirmed the necessity to adjust the RSM model constants. Escue and Cui (2010) simulated a swirling flow in a pipe using different RANS turbulence models. Their findings were similar to previous contributions in terms of turbulence model performance. The additional information from this work was that the RNG k-ε model performed better for $S < 1$, the RSM was better for $1 < S < 2$ and both models gave poor results for $S > 2$.

A further improvement of the accuracy is possible using Large Eddy Simulation LES and Direct Numerical Simulation DNS which became affordable to a certain extent for relatively large geometries and complex flows especially using LES. Hreiz et al. (2011) compared RANS, URANS and LES turbulence models for the single-phase simulation of a gas–liquid cylindrical cyclone GLCC. They found that LES also yields some discrepancies in the mean velocity profiles and turbulent kinetic energy. LES was seen to be superior in capturing the precessing vortex core PVC. Recently, CFD studies tend to use hybrid models. In terms of turbulence model, LES remains weak in the near wall regions unless the mesh is well refined to capture the vanishing turbulent scales in this region. The alternative is to use the Detached Eddy Simulation DES model which combines RANS and LES models. The RANS models are used in the near-wall region only (low Reynolds) while LES is used away from the walls. Javadi and Nilsson (2015) conducted a comparative study including different turbulence models with different degrees of complexity from the simple two equations models (RNG k-ε) to hybrid models (DDES). The hybrid models combined LES with the SST k-ω or Spalart–Allmaras URANS models. The hybrid models exhibited a clear superiority in predicting the vortex rope and its disintegration. LES studies, on swirling flows, do exist in the literature and there is a consensus that the classical sub-grid-scale SGS Smagorinsky model performs poorly in predicting swirling flows due, mainly, to the model constant which is not universal. Different sub-grid-scale SGS models were used in the literature such as the modified Smagorinsky model (Javadi and Nilsson, 2015), the localized dynamic procedure (Ranga Dinesh, 2007) and standard Smagorinsky model (Bulat et al., 2015) yielding acceptable degrees of accuracy.

To our knowledge, studies on the interaction of swirling flows with bluff bodies are scarce and were limited to spherical objects (e.g., Mattner et al., 2003; Atvars et al., 2009; Grinis and Tzadka, 2011). However, solid rods were used in cyclone separator applications, along the centerline of the device (e.g., Dyakowski and Williams, 1993), to alter the core flow totally or partially depending on the application to eliminate the air core for example and, eliminate particle re-entrainment consequently. In addition, the presence of a solid body at the center of the device would contribute in reducing the passage section to accelerate the flow and avoid a rapid swirl decay.

This work presents CFD results obtained from advanced LES simulations of a swirling pipe flow with and without the presence of a bluff body. The geometry considered mimics an important part of an in-line gas–liquid separator model based on a swirling flow interacting with a conical bluff body. The