CHEMICAL ENGINEERING RESEARCH AND DESIGN 102 (2015) 307–321

Contents lists available at ScienceDirect

Chemical Engineering Research and Design

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

# Effects of the inlet angle on the flow pattern and pressure drop of a cyclone with helical-roof inlet

*Dzmitry Misiulia*[*,1], *Anders Gustav Andersson*, *Tord Staffan Lundström*

*Division of Fluid and Experimental Mechanics, Department of Engineering Sciences and Mathematics,*
*Luleå University of Technology, SE-971 87 Luleå, Sweden*

ARTICLE INFO

*Article history:*
Received 13 May 2015
Received in revised form 18 June 2015
Accepted 28 June 2015
Available online 6 July 2015

*Keywords:*
Cyclone
Inlet angle
Flow pattern
Pressure drop
Large eddy simulation

ABSTRACT

The effects of inlet angle on the flow pattern and pressure drop in cyclones have been numerically investigated using Large Eddy Simulations with the dynamic Smagorinsky-Lilly subgrid-scale. Five cyclones with helical-roof inlets of different inlet angles and five cyclones with tangential inlets of different inlet heights at the same other geometric dimensions are considered. The results show that, increasing the inlet angle as well as the inlet height (inlet area) decreases the absolute values of positive (close to the cyclone wall) and negative (in the central region) static pressure and tangential velocity in the cyclone body that will probably reduce the collection efficiency. Also, increasing the inlet angle reduces the gas flow rates along the cyclone axis in both downward (outer) and upward (inner) vortices and increases the maximum radial velocity under the vortex finder that can enhance the number of small particles entrained by the gas flow and transferred from that region into the vortex finder and negatively affect the overall collection efficiency. The cyclone pressure drop is mainly generated by the losses in the cyclone body (under the vortex finder) and in the vortex finder. There is a significant decrease in pressure drop with increase of inlet angle. Based on the simulations an expression for the dimensionless pressure drop normalized by the inlet velocity for the cyclone with helical-roof inlet of different inlet angles is derived. Cyclones with helical-roof inlets have a higher aerodynamic efficiency as compared to cyclones with tangential inlets, and the highest aerodynamic efficiency was reached with an inlet angle of 20°.

## 1. Introduction

Cyclone separators are probably the most widely used devices for removing particles from gases, because of their relative ease of operation. Especially popular are reverse-flow cyclones with spiral, tangential and helical-roof inlet. The first cyclone patent (No. US 325521) was granted to John M. Finch of the United States back in 1885 and assigned to the Knickerbocker Company (Hoffmann and Stein, 2008). In Europe the first design of a cyclone was patented by the same company in Germany (No. DE 39219) in 1886. Since then, a lot of studies have been done to improve cyclone performance, i.e., pressure drop and separation efficiency, which are governed by many factors, among which cyclone geometry is one of the most important.

The inlet dimensions and configuration are among the most relevant parameters influencing the flow pattern within and performance of cyclones as reported in many articles. To

---

[*] *Corresponding author*. Postal address: Division of Fluid and Experimental Mechanics, Department of Engineering Sciences and Mathematics, Luleå University of Technology, SE-971 87 Luleå, Sweden. Tel.: +46 0 920 492392; fax: +46 0 920 491074.

E-mail addresses: dzmitry.misiulia@ltu.se, dzmitry.misiulia@gmail.com (D. Misiulia), anders.g.andersson@ltu.se (A.G. Andersson), staffan.lundstrom@ltu.se (T.S. Lundström).

[1] Permanent address: Department of Machines and Apparatus for Chemical and Silicate Production, Belarusian State Technological University, 13a Sverdlova str., 220006 Minsk, Belarus

http://dx.doi.org/10.1016/j.cherd.2015.06.036

exemplify, Elsayed and Lacor (2011a) showed that, increasing the inlet dimensions decreases the maximum tangential velocity in the cyclone and pressure drop and increases the cyclone cut-off diameter. They found that the effect of changing the inlet width is more significant than the inlet height especially for the cut-off diameter and the optimum ratio of inlet width $b$ to inlet height $a$ is somewhere between 0.5 and 0.7. Gimbun et al. (2005) showed that predicted pressure drop coefficients are proportional to the inlet area, which has also been confirmed experimentally by Hsiao et al. (2015). Hsiao et al. (2015) reported that reducing the inlet area under a constant operation flow rate increased the pressure drop and cut-off diameter while the pressure drop coefficient decreased. Their experimental results on examining the effect of the inlet aspect ratio ($a/b$) did not show a clear trend for the pressure drop coefficient, but the cut-off diameter decreased gradually with increasing $a/b$. However Iozia and Leith (1990) reported that the cyclone efficiency is independent of $a/b$, and the pressure drop of a cyclone with large $a/b$ is lower than that of a cyclone with a small $a/b$. Lidén and Gudmundsson (1997) argued that the cut-off size ought to be independent of the inlet geometry. Kenny and Gussman (2000) tested cyclones with different circular inlet model parts and suggested that the inlet was most important with respect to the effect of the cut-off diameter. Erdal and Shirazi (2006) reported that, the gradually reduced inlet nozzle geometry is the preferred geometry. Furthermore, the significant effects of the cyclone inlet dimensions on the cyclone performance have been reported by Movafaghian et al. (2000) and Avci and Karagoz (2001).

Some investigations have been done to derive the effect of inlet angle on cyclone performance with somewhat contradictory results. Funk et al. (2001) experimentally investigated cyclones with square and rectangular inlets at three inlet angles $\alpha$ ($-10°$, $0°$, $10°$). They found that square and angled inlet modifications lowered cyclone performance. Introducing dust–laden air into a cyclone at an upward angle of 10° such that it strikes the top of the cyclone reduced collection efficiency by 0.25%, from 0.9931 to 0.9906, while introducing the airflow downward did not reduce efficiency substantially from the efficiency realized with a horizontal inlet duct, the difference was 0.025%. Bernardo et al. (2005) numerically investigated a tangential cyclone with inlet angle of 45° and compared their results to experimental measurements of the cyclone with a conventional 0° angled inlet. They reported that increasing the inlet angle reduced the collection efficiency from 92% to 90.5% and increased pressure drop from 579 to 620 Pa. However later, Bernardo et al. (2006) computationally investigated an industrial-sized cyclone with a normal scroll 0° angled inlet and the three scroll inlet angles 30°, 45° and 60°. Their numerical results showed that increasing inlet section angle decreased the total pressure drop, while the overall collection efficiency increased for the cyclone with inlet section angle 30° and 45° and decreased for the cyclone with 60° inlet angle. Under the same operating conditions the predicted overall collection efficiency for the cyclone with $\alpha = 45°$ was 77.2%, while that for the normal inlet duct was 54.4%. Qian and Zhang (2007) numerically investigated the gas flow field of a tangential cyclone with three inlet section angles (0°, 30°, 45°) at the same inlet velocity (15 m/s) perpendicular to the inlet plane. They found that the inlet section angle increased the pressure drop of the cyclone whereas the pressure drop coefficients decreased. For calculating the pressure drop coefficients, they used the velocity in the inlet direction rather than the inlet velocity. However, the later investigation of Qian and Wu (2009) showed that with increasing the inlet section angle the cyclone pressure drop decreased and total separation efficiency increased. They reported that a 45° inlet section angle is the best option and it can reduce pressure drop of a cyclone by 15% and greatly increase the separation efficiency.

Also, the separation capability of a cyclone can be increased by using a multi-inlet configuration (DeOtte, 1990; Moore and McFarland, 1996; Gautam and Sreenath, 1997; Movafaghian et al., 2000; Lim et al., 2003; Zhao, 2005; Zhao et al., 2004, 2006; Yoshida et al., 2005, 2009; Martignoni et al., 2007).

In summary, despite the fact that a lot of work has been done to disclose the influence from inlet area, inlet aspect ratio, inlet section angle, number of inlets and the shape of the inlet section on the flow pattern and performance of a cyclone, the effects of inlet angle are not yet fully understood. Moreover, all articles mentioned above were focused on investigation of the inlet dimensions and configuration of cyclones with tangential or spiral inlets (with a flat roof), wherein inclined inlet duct a priori creates unfavorable conditions for fluid flow in the upper part of a cyclone body close to the roof. The flow either strikes the top of the cyclone at negative inlet angle or secondary flows appear there and a so-called dead zone is created at positive inlet angle. New studies are needed to fully understand the effect of inlet angle on the flow pattern and cyclone performance especially regarding cyclones with helical-roof inlet. Collection efficiency of a cyclone is largely determined by its aerodynamics. Therefore, the objective of the present study is to computationally investigate the effect of inlet angle on the flow pattern and pressure drop of a cyclone with helical-roof inlet using Computational Fluid Dynamics (CFD).

## 2. Numerical set-up

### 2.1. *Selection of the numerical model and governing equations*

The first numerical derivations of the flow field in a cyclone separator were probably performed by Boysan et al. (1982). From that time, CFD has been a successful method for performance estimations of cyclone separators. CFD is based on Navier-Stokes equations and turbulence is often modeled. However, the most accurate approach is direct numerical simulation (DNS) where the whole range of spatial and temporal scales of turbulence are resolved. The number of grid points required for fully resolved DNS is, however, enormously large, especially for high Reynolds number (Re) flows, and hence DNS is restricted to relatively low Re and limited geometries. An alternative approach is the large eddy simulation (LES) technique that is based on a separation between large and small turbulent scales. Scales that are of a characteristic size greater than the grid size are calculated directly and called large or resolved scales, and others are called small or subgrid-scales (SGS) and are modelled. LES is accepted as a promising numerical tool for solving the large-scale unsteady behavior of complex turbulent flows. The LES methodology has been used in many studies on highly swirling flow in cyclone separators (Derksen and Van den Akker, 2000; Slack et al., 2000; Shalaby et al., 2005; Lipowsky and Sommerfeld, 2007; Martignoni et al., 2007; Shalaby, 2007; Derksen et al., 2008; Shalaby et al., 2008; Elsayed, 2011; Elsayed and Lacor, 2011b, 2013; Misiulia et al., 2015).