Separation and Purification Technology 194 (2018) 470–479

Contents lists available at ScienceDirect

Separation and Purification Technology

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

Numerical analysis of a novel gas-liquid pre-separation cyclone

Long Huang[a,*], Songsheng Deng[a], Zhi Chen[b], Jinfa Guan[a], Ming Chen[a]

[a] Department of Military Oil Supply Engineering, Army Logistics University of PLA, Chongqing 401331, China
[b] China Petroleum Engineering Co., Ltd. Southwest Company, Chengdu 610041, China

ARTICLE INFO

Keywords:
Gas-liquid
Cyclone
Numerical simulation
Flow field
Separation performance

ABSTRACT

A novel gas-liquid cyclone characterized by the generation of swirl flow via guide vanes and a uniflow stream was designed for the pre-separation of a horizontal gravity based separator. The internal flow field and separation performance of the cyclone were investigated by numerical simulation. The Eulerian-Lagrangian approach with the Reynolds stress model (RSM) was used in the simulations. Contours of velocity and pressure within the cyclone are shown, and the trajectories of the droplets are also presented, demonstrating the separation mechanism of the cyclone. Then numerous simulations were conducted with different structural parameters to optimize the cyclone performance. The results show that broadening the width of the gap *lw* is good for large droplets separation; Decreasing the discharge angle α or increasing the torsion angle β of the guide vanes can increase the tangential velocity and improve the separation efficiency, but the pressure drop also increases fast; The increase of the diameter of the central body *Dc* will lead to an increase of tangential and axial velocity, and the pressure drop increases significantly as well.

1. Introduction

Cyclones are widely used devices for gas-solid or gas-liquid separation in various industries such as chemical, mineral, petroleum and environmental engineering [1,2]. There are several types of gas-liquid cyclones so far. The most widely used type is the Stairmand cyclone [1,3–5], which has a tangential inlet, a cylindrical section, a conical section and a vortex finder, as shown in Fig. 1(a). Another type is the gas-liquid cylindrical cyclone (GLCC) [6–9]. As shown in Fig. 1(b), a GLCC also has a tangential inlet, but its separation chamber is cylindrical. The third type has an axial inlet and the swirling flow is maintained by guide vanes, as shown in Fig. 1(c) [10–12].

However, the conventional gravity-based, vessel-type gas-liquid separator is still used in many fields due to its reliability and management convenience. But the separation performance of the gravity-based separator does need to be improved. Thus the idea of combining different separation techniques was promoted.

A new gas-liquid separator was designed. Its main body is a horizontal gravity based separator, but what differs from the conventional ones is that a novel cyclone separator is mounted in the inlet for pre-separation, as shown in Fig. 2. The gas-liquid stream enters the separator through the cyclone, and voluminous droplets are first separated by the novel cyclone. The structure and size of the newly designed cyclone should match the limited space in the inlet of the horizontal separator, so the tangential inlet designs (Stairmand cyclone and GLCC) are not suitable. Therefore, the cyclone is designed on the base of guide vanes cyclone as shown in Fig. 1(c). And the gas-liquid stream in the cyclone should goes straight through the cyclone rather than reverse, because there is no space for the complex structure such as vortex finder and branch pipe. Thus a novel pre-separation cyclone was designed. The feed enters the novel cyclone axially, and then the stream is made to spin by guide vanes. Most droplets are separated by centrifugal force and the rest droplets are discharged into the horizontal gravity separator for further separation by gravity. As shown in Fig. 2, the outlet of the cyclone is an elbow which makes the discharged gas stream blow toward the wall of the horizontal separator. This design can avoid the gas flow to directly impact on the liquid surfaces which may carry off droplets from liquid; moreover, it can separate droplets via the collision of the flow with the wall of the horizontal separator. The original design is used for shale gas production [13–15], and it can also be used for other industries for gas purification.

With the development of computer technology and progresses in multiphase flows, computational fluid dynamics (CFD) has been used more and more in the studies of cyclones. Compared with experimental method, CFD can reduce the cost of time and money in the design process and provide much more details of the flow field and particle dynamics inside the cyclone. Many researchers have visualized the flow pattern in cyclones and studied the performance of cyclone separators through CFD [2,9,12,16–18]. Another important role of CFD is to improve the performance of cyclones by evaluating the effects of

* Corresponding author at: Department of Military Oil Supply Engineering, Army Logistics University of PLA, Huxi University Town, Shapingba District, Chongqing 401331, PR China.
E-mail address: hlong0000@163.com (L. Huang).

https://doi.org/10.1016/j.seppur.2017.11.066
Received 4 August 2017; Received in revised form 19 October 2017; Accepted 26 November 2017
1383-5866/

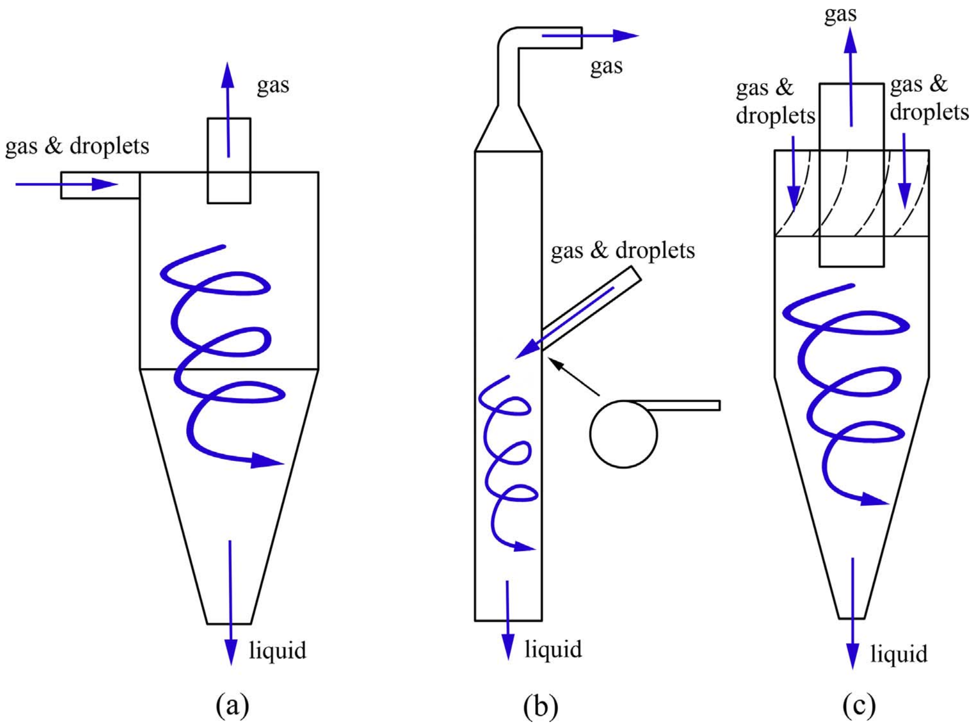

Fig. 1. Diagram of different types of gas-liquid cyclone.

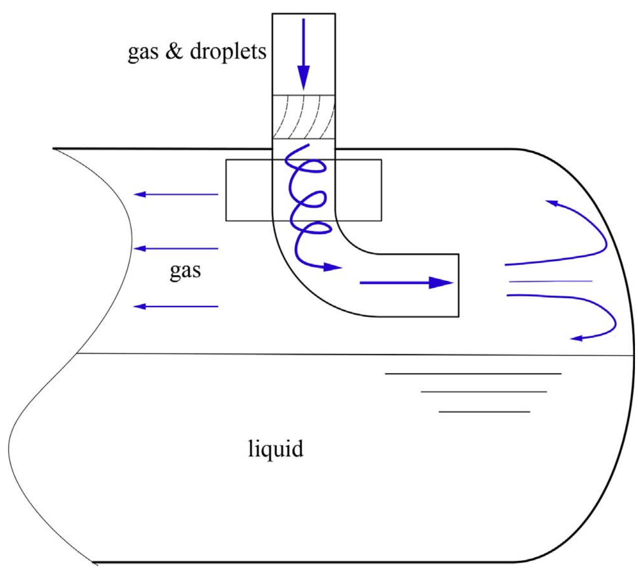

Fig. 2. Sketch of the new horizontal gas-liquid separator.

geometric and operational parameters. Lots of numerical simulations have been conducted to optimize the structure of cyclones, such as the cyclone size [19], vortex finder [20,21], inlet [22,23] and outlet [24].

Thus, this study is intended to obtain detailed flow information in the novel pre-separation cyclone, and to improve the performance of the cyclone by evaluating the effects of geometric parameters through CFD simulation. Velocity distributions, pressure contour, and droplet trajectories within the cyclone are shown, and better understanding of the separation process in this cyclone is provided.

2. Structure of the pre-separation cyclone

As shown in Fig. 3(a), the structure of the pre-separation cyclone is designed for assembling with the inlet of the gravity based separator. It has an axial inlet with a cylindrical shape whose external diameter matches the inner diameter of the horizontal separator's inlet. The cyclone mainly consists of a central body on which guide vanes are mounted, an annular gap from which droplets are threw out, a liquid reservoir, and an elbow outlet. The characteristic dimensions are also shown in Fig. 3 and the details could be found in Table 1.

The working principle of the cyclone is as follows. The gas-liquid mixture enters cyclone axially, and becomes a swirling flow when it flow through the guide vanes. Under the centrifugal force, the droplets are thrown to the wall and enter the liquid reservoir through the annular gap. The gas and the residual droplets exit the cyclone via the outlet and enter the horizontal separator for further separation.

3. Numerical methods

3.1. Continuous phase modeling

For cyclones, the pressure and temperature changed little in the separation process, so the gas flow in the cyclone was assumed as incompressible [3]. Many researchers used Reynolds average Navier–-Stokes (RANS) model to predict the flow pattern in cyclones, and it had been proven that the Reynolds stress model (RSM) was a good choice for the simulation of the swirling flow in cyclones, which can provide accurate predictions [25,26]. So RSM was used in this study for turbulence modeling of the continuous phase.

The continuity and Reynolds average Navier-Stokes equations are given as:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0. \tag{1}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu\frac{\partial^2 \overline{u_i}}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j}\overline{u_i' u_j'} \tag{2}$$

where $\overline{u_i}$ is mean velocity; ρ is the density; ν is the kinematic viscosity; $\overline{p}$ is mean pressure; $\overline{u_i' u_j'}$ represents Reynolds stress tensor; here $u_i' = u_i - \overline{u_i}$ is the fluctuating velocity.

In order to close the equations, $\overline{u_i' u_j'}$ must be modeled. In RSM, the turbulence transport equation is as follows [21]:

$$\frac{\partial}{\partial t}(\rho\overline{u_i' u_j'}) + \frac{\partial}{\partial x_k}(\rho u_k \overline{u_i' u_j'}) = D_{T,ij} + P_{ij} + \varphi_{ij} + \varepsilon_{ij} + S \tag{3}$$