Technical University of Denmark

Gas-liquid two-phase flows in double inlet cyclones for natural gas separation

Yang, Yan; Wang, Shuli; Wen, Chuang

Published in:
Cogent Engineering

Link to article, DOI:
10.1080/23311916.2017.1373421

Publication date:
2017

Document Version
Publisher's PDF, also known as Version of record

Citation (APA):
Yang, Y., Wang, S., & Wen, C. (2017). Gas-liquid two-phase flows in double inlet cyclones for natural gas separation. Cogent Engineering, 4(1), [1373421]. DOI: 10.1080/23311916.2017.1373421

MECHANICAL ENGINEERING | RESEARCH ARTICLE

Gas-liquid two-phase flows in double inlet cyclones for natural gas separation

Yan Yang[1,2], Shuli Wang[1] and Chuang Wen[2*]

Received: 19 July 2017
Accepted: 25 August 2017
First Published: 31 August 2017

*Corresponding author: Chuang Wen, Department of Mechanical Engineering, Technical University of Denmark, Nils Koppels Allé, 2800 Kgs. Lyngby, Denmark
E-mail: cwen@mek.dtu.dk.

Revieiwng editor:
Duc Pham, University of Birmingham, UK

Additional information is available at the end of the article

Abstract: The gas-liquid two-phase flow within a double inlet cyclone for natural gas separation was numerically simulated using the discrete phase model. The numerical approach was validated with the experimental data, and the comparison results agreed well with each other. The simulation results showed that the strong swirling flow produced a high centrifugal force to remove the particles from the gas mixture. The larger particles moved downward on the internal surface and were removed due to the outer vortex near the wall. Most of the tiny particles went into the inner vortex zones and escaped from the up-outlet. The swirling flow was concentric due to the design of the double inlet for the cyclonic separator, which greatly improved the separating efficiency. The separating efficiency was greater than 90% with the particle diameter of more than 100 μm.

Subjects: Energy & Fuels; Technology; Novel Technologies

Keywords: cyclone separation; discrete phase model (DPM); separating efficiency; gas dehydration

ABOUT THE AUTHORS

Yan Yang holds a Ph.D. degree in Oil and Gas Engineering from China University of Petroleum. Dr. Yang has published over 40 peer-reviewed journal and conference publications in the fields of corrosion and multiphase flows.

Shuli Wang is Professor Emeritus of Oil and Gas Engineering at Changzhou University. Dr. Wang has over 35 years of teaching and research experience with authorship of more than 100 research publications.

Chuang Wen has multidisciplinary computational and experimental experience in a broad area of fluid dynamics. He is focusing on the development and application of the computational fluid dynamics for the complicated multiphase flow system, which aims to provide an efficient way for the cleaning utilization and sustainable development of the energy system.

The present paper proposes a methodology for the cleaning utilization of natural gas by employing a double inlet cyclone separator to improve the collection efficiency of the gas-liquid two-phase flows.

PUBLIC INTEREST STATEMENT

The demand for energy supply is increasing continuously in the last decades, in which natural gas plays a significant strategic role with approximately 24% of primary energy consumption. The clean utilization of natural has been recognised as one of the most promising measures to improve the sustainable development of fossil energy. However, the safety operation is a critical issue during the production, transportation, storage and processing of natural gas. The purpose of the current study is to propose a methodology to remove the liquids from natural gas using a double inlet cyclone separator. The discrete phase model and Reynolds stress model are employed to predict the gas-liquid two-phase flow behavior. The numerical results indicate that the swirling flow was concentric due to the design of the double inlet for the cyclonic separator, which greatly improved the separating efficiency.

1. Introduction

Natural gas is relatively a clean source of energy, and its demand is increasing rapidly with industrialization. When natural gas is extracted from wells, it contains a small fraction of moisture in the form of water vapour. When moving up to the surface from the well, the moisture contained by the gas condensates that may cause some serious issues. For instance, it may induce the hydrate formation to block the pipelines, and accelerate the corrosion problem (Wen, Li, Wang, & Yang, 2015; Yang, Li, & Wen, 2013; Yang, Wang, & Wen, 2016). Therefore, the water droplets need to be separated out from natural gas by some means to protect the pipelines from corrosion and hydrate formation.

The cyclones are widely used in many industrial areas for a dense phase removal from the multiphase flows (Cortes & Gil, 2007). Over the years, the Lee type cyclones (Kim & Lee, 1990), the semi-spherical cyclones (Ogawa, Hironaka, Kato, & Seito, 1991), the PV type cyclones (Chen, Sun, & Shi, 2001), the reverse flow cyclones (Sun, Chen, & Shi, 2005; Tien & Ray, 2000), the dynamic cyclones (Jiao, Zheng, Wang, & Sun, 2008), and the square cyclones (Raoufi, Shams, & Kanani, 2009; Su & Mao, 2006), have been designed to meet the demands. To design and optimize the cyclonic separators, various methods have been employed to characterize the separation performance, including the theoretical analysis (Avci & Karagoz, 2001; Chmielniak & Bryczkowski, 2000; Enliang & Yingmin, 1989), numerical simulation (Chu, Wang, Xu, Chen, & Yu, 2011; Griffiths & Boysan, 1996; Oh, Choi, & Kim, 2015; Sgrott, Noriler, Wiggers, & Meier, 2015) and experimental test (Baltrėnas & Chlebnikovas, 2015; Hoekstra, Derksen, & Van Den Akker, 1999; Ji, Xiong, Wu, Chen, & Wu, 2009). Computational Fluid Dynamics (CFD) is an effective tool to investigate and develop the cyclones for its reducing design cycle and costly experimental costs. The gas-liquid two-phase flow in a cyclone can be simulated using a two-phase flow model. In these models, Eulerian methods are usually employed to deal with the continuous phase, while either Eulerian or Lagrangian method is generally utilized to simulate the discrete phase. Compared with the Euler-Euler model, the Lagrangian approach, can describe the motion of a single particle in detail, and correspondingly obtain the separation efficiency for the cyclone separators.

In this paper, we focus on the low-liquid loading in natural gas production, and a revised cyclone separator was employed for the removal of the free water. The CFD was employed to predict the separation performance of a double inlet cyclone for natural gas dehydration with water content below 10%. The natural gas flow fields and liquid particles in the cyclone were calculated using the discrete phase model (DPM) and Reynolds stress model.

2. Computational model

2.1. Continuous phase governing equations

In the cyclone separators, the gas continuous can be discretized by the mass and momentum conservation equations.

The mass equation can be described in Equation (1).

$$\frac{\partial \rho_g}{\partial t} + \nabla \cdot (\rho_g u_g) = 0 \tag{1}$$

where, ρ_g is the gas density; u_g is the gas velocity; t is time.

The momentum equation is shown as follows:

$$\frac{\partial}{\partial t}(\rho_g u_g) + \nabla \cdot (\rho_g u_g u_g) = -\nabla p + \nabla \Gamma + F \tag{2}$$

where p, Γ, F are, respectively, the gas phase pressure, viscous stress tensor and gas-particle interaction force. The viscous stress tensor Γ in Equation (2) is given in (Bird, Warren, & Edwin, 1960):

$$\Gamma = -\frac{2}{3}\mu_g \nabla \cdot u_g \delta_k + \mu_g \left[\left(\nabla \cdot u_g \right) + \left(\nabla \cdot u_g \right)^{-1} \right] \tag{3}$$

where μ_g and δ_k are the gas viscosity and the Kroneker delta. The gas-particle interaction force will be discussed in the Section 2.3.

2.2. Discrete phase governing equations

A particle motion in a double inlets cyclone is described as translation and rotation calculated with Newton's second law of motion. The one-way coupling method was used to simulate the discrete phase. The droplet agglomeration and break-off on the walls were not considered in this simulation case. The governing equation for a discrete liquid particle is:

$$m_l \frac{du_l}{dt} = m_l g - \rho_g g V_l + F \tag{4}$$

where m_l, $\boldsymbol{u_l}$ and V_l are, respectively, the mass, velocity and volume of the liquid particle. In a cyclone, the gravity force plays a significant role to separate the liquid from the natural gas. Therefore, the gravity force is considered in our calculation. The buoyancy force was ignored due to the huge difference in density between the gas and liquid phases.

2.3. Coupling between continuous and discrete phases

The model for the discrete liquid particles was at an individual particle level, whereas that for a gas phase was at a computational cell level. Therefore, it needs to couple the numerical scales. This is actually achieved by considering the gas-particle interaction force $\boldsymbol{F}$. In a cyclone separator, the gas-particle interaction force includes the drag force $\boldsymbol{F_d}$ and the saffman force $\boldsymbol{F_s}$. That is:

$$\boldsymbol{F} = \boldsymbol{F_d} + \boldsymbol{F_s} \tag{5}$$

The drag force is given in (Ounis, Ahmadi, & McLaughlin, 1991):

$$\boldsymbol{F_d} = \frac{18\mu_g}{\rho_l d_l^2} \frac{C_D R_e}{24} (\boldsymbol{u_g} - \boldsymbol{u_l}) \tag{6}$$

where ρ_l and d_l are the density and diameter of the liquid particle, respectively. The factor C_c is the Cunningham correction to Stokes' drag law.

The Saffman's lift force, is provided by Saffman (Saffman, 1965):

$$\boldsymbol{F_s} = \frac{2K\sqrt{u_g}\rho d_{ij}}{\rho_l d_l (\rho_{lk} d_{kl})^{0.25}} (\boldsymbol{u_g} - \boldsymbol{u_l}) \tag{7}$$

where d_{ij} is the deformation tensor $K = 2.594$.

2.4. Geometry and mesh generation

For a cyclone separator, the inlet configuration is an important factor to affect the flow pattern and particle separation performance. Many studies were conducted on this topic and the finds show that a double inlet cyclone was preferred over a single inlet model. Zhao, Su, and Zhang (2006) numerically investigated the single and spiral double inlet cyclones by evaluating the gas flow field and particle collection efficiency. They found that the double inlet cyclone showed a better performance on the symmetry of vortex core than the single inlet model. Martignoni, Bernardo, and Quintani (2007) studied the effect of three kinds of cyclones on the separation performance with the Reynolds stress model and large eddy simulation, including the single inlet, double inlet, and volute inlet and outlet models. The simulation results on the total pressure drop and separation efficiency showed that the double inlet cyclone was a better choice than the inlet one. Yoshida, Yoshikawa, Fukui, and Yamamoto (2008) evaluated the effect of the inlet geometry on the particle separation performance of the hydro-cyclones. The numerical and experimental results showed that the double inlet cyclone obtained the uniform distribution of the fluid velocity and high collection efficiency compared to the single inlet one.

Base on the above mention studies and their findings, a double inlet cyclone separator was employed to remove the free water from natural gas, in which the cylinder type of atomizer was mounted vertically in the cyclone separator. This modified cyclone used in this study included double inlets, a vortex finder, a separation chamber, a conical section and water hopper, as shown in Figure 1. The tangential double inlets were preferred to separate the water particles and improve the separating efficiency. The mesh for the cyclone separator was generated with the tetrahedral grids for the circular inlets area and hexahedral for other parts. The grid independence of grid size was investigated with the coarse (367,626), medium (628, 463), and fine (1, 216, 761) cells. The total pressure drop was used as the characteristic parameter, and the results were shown in Table 1. The relative error, *RE*, is defined as $RE = |P_g - P_f|/P_f * 100\%$, where the P_g is the total pressure drop from the computing mesh, P_f is the total pressure drop from the fine mesh. The relative error between the medium and fine grids was only about 4.83%, and the final computational domain contains 628, 463 control mesh cells.

Figure 1. Geometry of the double inlet cyclone.

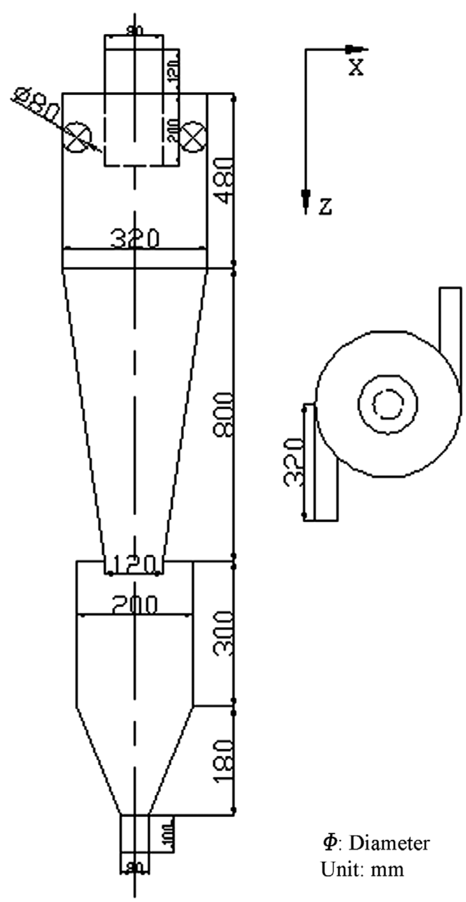


Table 1. Grid independency test		
Grid numbers	**Total pressure drop (Pa)**	**Relative error (%)**
367,626	224	8.21
628, 463	217	4.83
1, 216,761	207	0

2.5. Numerical scheme

The ANSYS FLUENT software was used as a computational platform for our numerical simulation. The SIMPLE was employed for the pressure-velocity coupling. The different turbulence models can be applied to predict the gas-liquid two flows, including k-ε model, k-ω model, and Reynolds stress model (Wen, Cao, Yang, & Li, 2012; Yang, Li, & Wen, 2017; Yang, Walther, Yan, & Wen, 2017; Yang & Wen, 2017). The Reynolds stress model was used to model the high swirling flow in the cyclone separators. The continuity equation, momentum equation and turbulence equations were discretized with the QUICK method. The second order upwind was adopted to calculate the turbulent kinetic energy, turbulent dissipation rate, and Reynolds stress.

2.6. Boundary conditions

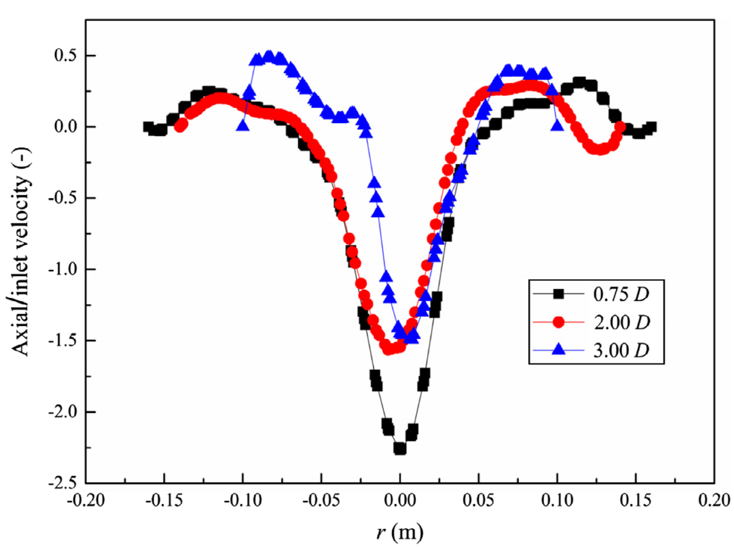
In the cyclone separator, the mass flow rate was used for the inlet, when the outlet vent and out flow were utilized for the gas and particle outlets, respectively. The inlet velocity is assigned to 10 m/s. For the discrete phase, the escape was set for the inlet and gas outlet. It means that a particle cannot return the computational domain, if they reach the inlet due to the back flow. The trap boundary conditions are utilized for the liquid outlet and the walls of the conical section and water hopper. The reflect condition was appointed for all other walls. In our numerical cases, it was assumed that the particle was sphere liquid of water with the density of 998.2 kg/m^3. The liquid particles were injected along with the gas phase. The mass flow rate of the water liquids was about 0.8 kg/s for the simulation.

3. Results and discussion

3.1. Gas axial and tangential velocity

The validation of the continuous phase has been conducted by Zhao et al. (2006), and the results showed that the CFD model could predict the gas flow field in a double inlet cyclone with a high accuracy. Therefore, we did not perform the repetitive work and showed our simulation results directly. Figure 2 shows the axial velocity at the representative sections, z = 0.75 D, 2.00 D, 3.00 D, which represent the flow distribution at the typical section of the double inlet cyclone separator. z = 0.75 D locates at the middle of the top cylindrical section. z = 2.0 D locates at the beginning of the cone section. z = 3.0 D locates at the 60% of the cone section. The negative axial velocity of gas phase was observed in the central region, which indicated the aggregation of natural gas to the core area. Most of the gas exhausted from the gas outlet in the cyclone separator. However, the gas axial velocity was positive in the outer zones. In this condition, natural gas went down along the inner surface, and correspondingly a small quantity of gas escaped from the liquid outlet. In other words, the dry gas

Figure 2. Axial velocity at the typical sections.



went through the gas outlet, while the water liquids moved down to the liquid outlet due to the centrifugal force and gravity.

The tangential component of the gas velocity at some typical cross sections is illustrated in Figure 3. It showed that in these cross sections, the tangential velocity increased rapidly to a maximum firstly and then decreased to about 2 m/s in the central region. The swirling flow fields were divided into the inner and outer vortex zones by the maximum tangential velocity. The weak inner swirls

Figure 3. Tangential velocity at the typical sections.

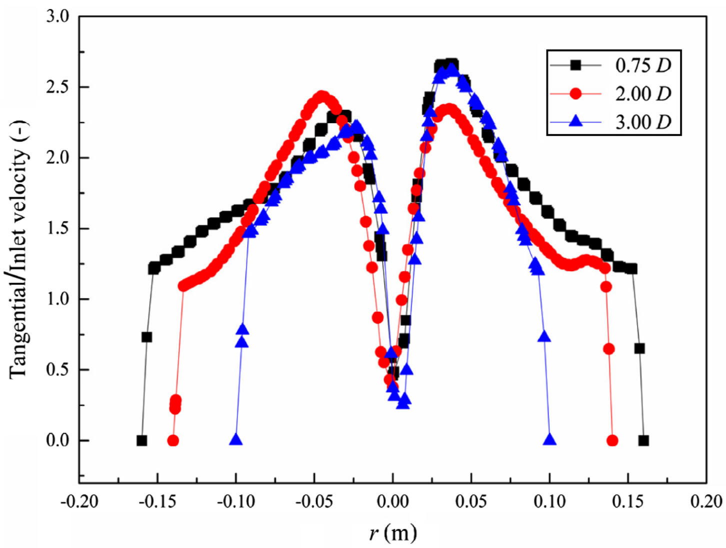
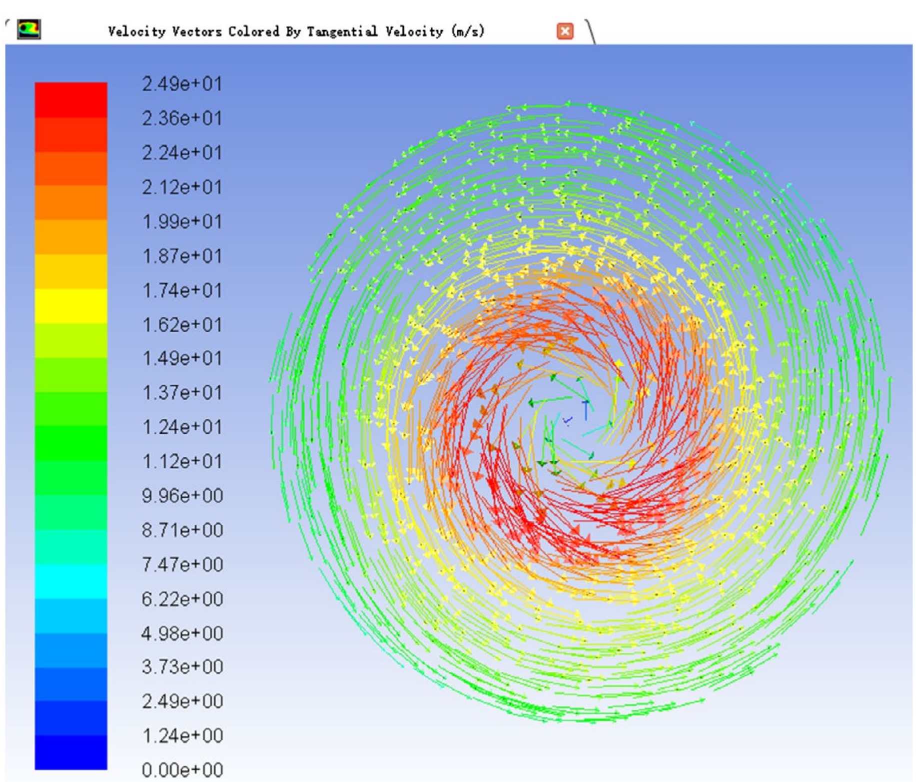


Figure 4. Vector of the gas flow field.



made the dry gas stay in the central region. The water particles were centrifuged to the wall owing to the strong outer vortex.

3.2. Vector of gas flow

Figure 4 shows the flow vector of the gas flow field at $z = 2.00\,D$. A vortex represented in the cyclonic separator, and correspondingly a high centrifugal force was generated to remove the liquids from the gas mixture. The pseudo-free vortex generated a weaker centrifugal force and the larger particles went downward as a result of the gravity. It indicates that the double inlet separator forms the concentric distribution of the swirling flow, which significantly improves the separating efficiency of the cyclonic separator.

3.3. Particle trajectories

In this section, we utilized the DPM model to predict the motion characteristics of the liquid particles. In this simulation, the assumptions were as follows: (a) each particle was a sphere, (b) the initial velocity of the particles was equal to the gas speed, (c) the liquids were removed as soon as they are captured at the liquid outlet, (d) the range of the size of particle diameter is taken random numbers from 5 to 300 μm, which cover the range for the water contained in the natural gas.

The particle trajectories in the cyclone are shown in Figure 5. We released and tracked the particles at the double inlet. It could be seen that the particles moved in a circular motion under the

Figure 5. Liquid particle trajectories in the cyclone. (a) Minor diameter of particles inlet and outlet: 5 μm, (b) Moderate diameter of particles inlet and outlet: 50 μm, and (c) Major diameter of particles inlet and outlet: 100 μm.

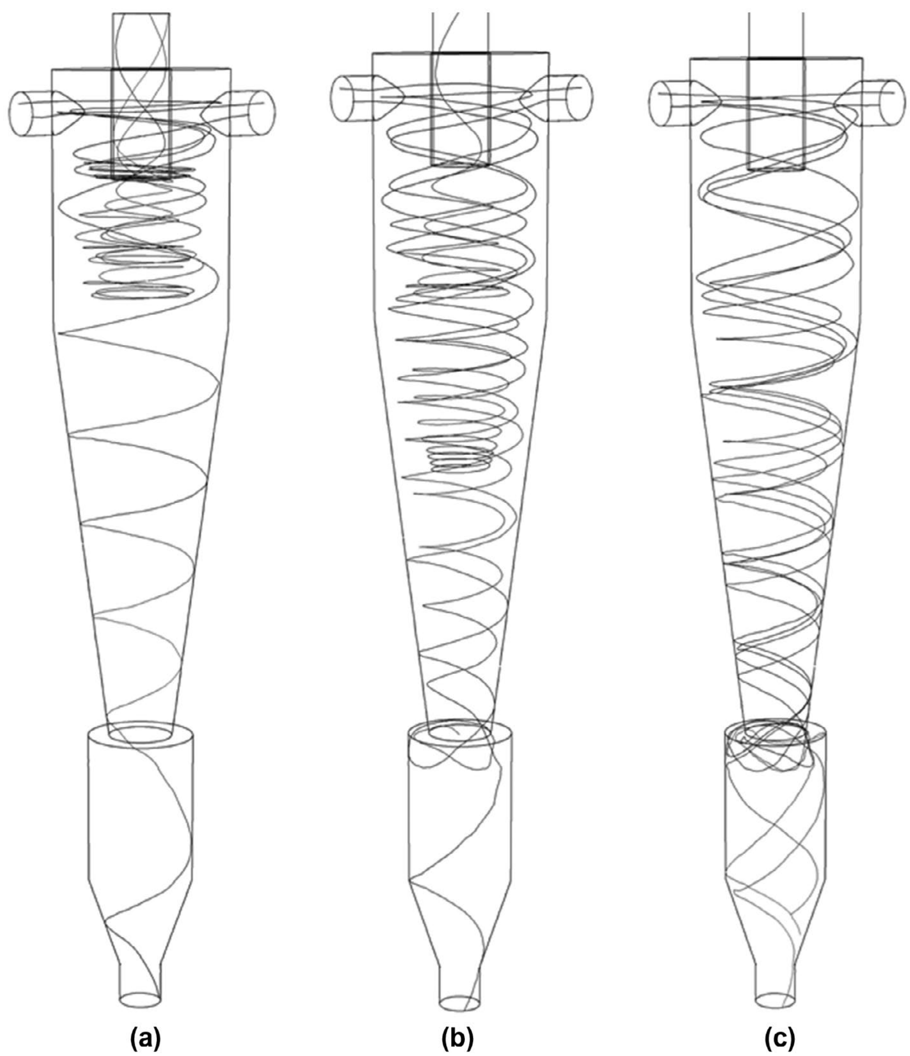
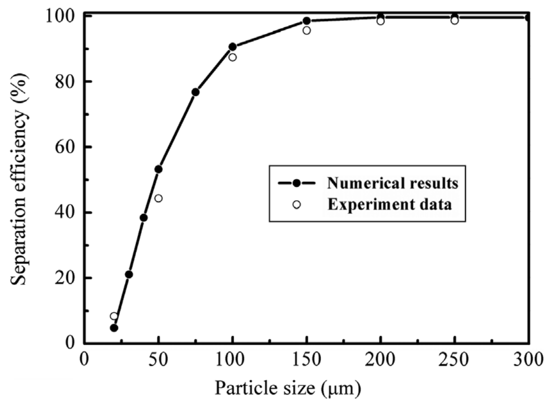


Figure 6. Comparison of numerical and experimental separating efficiency.



combined effects of the drag, centrifugal and Saffman forces. These particle trajectories were consistent with the findings obtained by other investigators (Gimbun, Chuah, Choong, & Fakhru'l-Razi, 2005; Shukla, Shukla, & Ghosh, 2011; Wang, Xu, Chu, & Yu, 2006). For the particle with a minor diameter, e.g. d = 5 μm, most of these particles went into the inner vortex zones and escaped from the up-outlet when they were released in the inlet, while only very few particles moved down to the liquid outlet. For the particle with a moderate diameter, e.g. d = 50 μm, some of the particles collided with the wall under the effect of the outer vortex and finally passed through the liquid outlet. The others moved into the inner vortex zones, and the particles went downward some distance and then escaped from the up-outlet along with this swirling flow. When the particles became big enough, such as d = 100 μm, almost all the particles would go with the outer vortex flow, and moved down to the liquid outlet under the action of gravity.

3.4. Separating efficiency

The separating efficiency of the liquid particles obtained by numerical simulation and experimental data are shown in Figure 6. The experiment was conducted in an indoor pipeline loop system. The compressed air was produced by a screw compressor, the gas flow rate was controlled by an electric control valve. The gas pressure was measured by the Rosemount3051 type pressure sensors with a precision of ± 0.075%. A vortex flow meters was employed to measure the gas flow rate with an accuracy grade ± 1.0% (Yokogawa Electric Corporation, 2007). The free water was injected by an atomizing nozzle after filtered and purified. It could be observed that when the particle diameter was about 30 μm, the separating efficiency was below 20%. The separating efficiency increased rapidly when the diameter of the liquid particle changed from 50 to 300 μm. If the particle diameter was greater than 100 μm, the separating efficiency was greater than 90%. The numerical results showed a reasonable agreement with the experimental data. It demonstrated that the DPM and Reynolds stress model could effectively predict the gas-liquid separation in the double inlet cyclones.

4. Conclusions

The Reynolds stress model and discrete phase model were employed to simulate the natural gas flows in the double inlet cyclone. The results showed that the swirling flow was concentric due to the design of the double inlet, significantly improving the separating efficiency of the cyclonic separator. The numerical results were in good agreements with the experimental data. The computational fluid dynamics provides a fundamental approach to analyze the gas-liquid separation characteristics in the cyclones. It should be noted that the separating efficiency can be changed depending upon the cone angle of injected water and the injection velocity. These should be further studied in the future work to improve the performance of designed double inlet cyclone separators.

Funding
This work was supported by the National Natural Science Foundation of China [grant number 51574045]. The research leading to these results has received funding from the People Programme (Marie Curie Actions) of the European Union's Seventh Framework Programme [grant number FP7/2007-2013]; the REA [grant number 609405] (COFUNDPostdocDTU).

Author details
Yan Yang[1,2]
E-mail: yyan-petroleum@cczu.edu.cn
ORCID ID: http://orcid.org/0000-0001-5796-7990
Shuli Wang[1]
E-mail: wsl@cczu.edu.cn
Chuang Wen[2]
E-mail: cwen@mek.dtu.dk
ORCID ID: http://orcid.org/0000-0002-4445-1589
[1] School of Petroleum Engineering, Changzhou University, Zhonglou District, Changzhou, Jiangsu 213016, China.
[2] Department of Mechanical Engineering, Technical University of Denmark, Nils Koppels Allé, 2800 Kgs. Lyngby, Denmark.

Citation information
Cite this article as: Gas-liquid two-phase flows in double inlet cyclones for natural gas separation, Yan Yang, Shuli Wang & Chuang Wen, *Cogent Engineering* (2017), 4: 1373421.

References

Avci, A., & Karagoz, I. (2001). Theoretical investigation of pressure losses in cyclone separators. *International Communications in Heat and Mass Transfer, 28*, 107–117. https://doi.org/10.1016/S0735-1933(01)00218-4

Baltrėnas, P., & Chlebnikovas, A. (2015). Experimental research on the dynamics of airflow parameters in a six-channel cyclone-separator. *Powder Technology, 283*, 328–333. https://doi.org/10.1016/j.powtec.2015.06.005

Bird, R. B., Warren, E. S., & Edwin, E. L. (1960). *Transport phenomena*. NewYork, NY: Wiley.

Chen, J., Sun, G., & Shi, M. (2001). An experimental research on the pilot-scale cold model of the three-stage PV type cyclone separator for acrylonitrile reactors. *Chemical Engineering & Machinery, 5*, 187–192.

Chmielniak, T., & Bryczkowski, A. (2000). Method of calculation of new cyclone-type separator with swirling baffle and bottom take off of clean gas–part I: Theoretical approach. *Chemical Engineering and Processing: Process Intensification, 39*, 441–448. https://doi.org/10.1016/S0255-2701(00)00094-5

Chu, K. W., Wang, B., Xu, D. L., Chen, Y. X., & Yu, A. B. (2011). CFD–DEM simulation of the gas–solid flow in a cyclone separator. *Chemical Engineering Science, 66*, 834–847. https://doi.org/10.1016/j.ces.2010.11.026

Cortes, C., & Gil, A. (2007). Modeling the gas and particle flow inside cyclone separators. *Progress in Energy and Combustion Science, 33*, 409–452. https://doi.org/10.1016/j.pecs.2007.02.001

Enliang, L., & Yingmin, W. (1989). A new collection theory of cyclone separators. *AIChE Journal, 35*, 666–669. https://doi.org/10.1002/(ISSN)1547-5905

Gimbun, J., Chuah, T. G., Choong, T. S., & Fakhru'l-Razi, A. (2005). A CFD study on the prediction of cyclone collection efficiency. *International Journal for Computational Methods in Engineering Science and Mechanics, 6*, 161–168. https://doi.org/10.1080/15502280590923649

Griffiths, W. D., & Boysan, F. (1996). Computational fluid dynamics (CFD) and empirical modelling of the performance of a number of cyclone samplers. *Journal of Aerosol Science, 27*, 281–304. https://doi.org/10.1016/0021-8502(95)00549-8

Hoekstra, A. J., Derksen, J. J., & Van Den Akker, H. E. A. (1999). An experimental and numerical study of turbulent swirling flow in gas cyclones. *Chemical Engineering Science, 54*, 2055–2065. https://doi.org/10.1016/S0009-2509(98)00373-X

Ji, Z., Xiong, Z., Wu, X., Chen, H., & Wu, H. (2009). Experimental investigations on a cyclone separator performance at an extremely low particle concentration. *Powder Technology, 191*, 254–259. https://doi.org/10.1016/j.powtec.2008.10.015

Jiao, J., Zheng, Y., Wang, J., & Sun, G. (2008). Experimental and numerical investigations of a dynamic cyclone with a rotary impeller. *Chemical Engineering and Processing: Process Intensification, 47*, 1861–1866. https://doi.org/10.1016/j.cep.2007.10.015

Kim, J., & Lee, K. (1990). Experimental study of particle collection by small cyclones. *Aerosol Science and Technology, 12*, 1003–1015. https://doi.org/10.1080/02786829008959410

Martignoni, W. P., Bernardo, S., & Quintani, C. L. (2007). Evaluation of cyclone geometry and its influence on performance parameters by computational fluid dynamics (CFD). *Brazilian Journal of Chemical Engineering, 24*, 83–94. https://doi.org/10.1590/S0104-66322007000100008

Ogawa, A., Hironaka, A., Kato, T., & Seito, O. (1991). Velocity distributions of gas flow and the separation efficiency of the semi-spherical cyclone. *Advanced Powder Technology, 2*, 191–212. https://doi.org/10.1163/156855291X00288

Oh, J., Choi, S., & Kim, J. (2015). Numerical simulation of an internal flow field in a uniflow cyclone separator. *Powder Technology, 274*, 135–145. https://doi.org/10.1016/j.powtec.2015.01.015

Ounis, H., Ahmadi, G., & McLaughlin, J. B. (1991). Brownian diffusion of sub-micrometer particles in the viscous sub-layer. *Journal of Colloid and Interface Science, 143*, 266–277. https://doi.org/10.1016/0021-9797(91)90458-K

Raoufi, A., Shams, M., & Kanani, H. (2009). CFD analysis of flow field in square cyclones. *Powder Technology, 191*, 349–357. https://doi.org/10.1016/j.powtec.2008.11.007

Saffman, P. G. (1965). The lift on a small sphere in a slow shear flow. *Journal of Fluid Mechanics, 22*, 385–400. https://doi.org/10.1017/S0022112065000824

Sgrott, O. L., Noriler, D., Wiggers, V. R., & Meier, H. F. (2015). Cyclone optimization by COMPLEX method and CFD simulation. *Powder Technology, 277*, 11–21. https://doi.org/10.1016/j.powtec.2015.02.039

Shukla, S. K., Shukla, P., & Ghosh, P. (2011). Evaluation of numerical schemes for dispersed phase modeling of cyclone separators. *Engineering Applications of Computational Fluid Mechanics, 5*, 235–246. https://doi.org/10.1080/19942060.2011.11015367

Su, Y., & Mao, Y. (2006). Experimental study on the gas–solid suspension flow in a square cyclone separator. *Chemical Engineering Journal, 121*, 51–58. https://doi.org/10.1016/j.cej.2006.04.008

Sun, G., Chen, J., & Shi, M. (2005). Optimization and applications of reverse-flow cyclones. *China Particuology, 3*, 43–46. https://doi.org/10.1016/S1672-2515(07)60162-6

Tien, Y., & Ray, M. (2000). Development of a post cyclone to improve the efficiency of reverse flow cyclones. *Powder Technology, 113*, 97–108.

Wang, B., Xu, D. L., Chu, K. W., & Yu, A. B. (2006). Numerical study of gas–solid flow in a cyclone separator. *Applied Mathematical Modelling, 30*, 1326–1342. https://doi.org/10.1016/j.apm.2006.03.011

Wen, C., Cao, X., Yang, Y., & Li, W. (2012). An unconventional supersonic liquefied technology for natural gas. *Energy*

Education Science and Technology Part A: Energy Science and Research, 30, 651–660.

Wen, C., Li, J., Wang, S., & Yang, Y. (2015). Experimental study on stray current corrosion of coated pipeline steel. *Journal of Natural Gas Science and Engineering, 27*, 1555–1561. https://doi.org/10.1016/j.jngse.2015.10.022

Yokogawa Electric Corporation. (2007). *Digital Vortex Flowmeter digitalYEWFLO*. Retrieved from https://www.yokogawa.com

Yang, Y., Li, A., & Wen, C. (2017). Optimization of static vanes in a supersonic separator for gas purification. *Fuel Processing Technology, 156*, 265–270. https://doi.org/10.1016/j.fuproc.2016.09.006

Yang, Y., Li, Z., & Wen, C. (2013). Effects of alternating current on X70 steel morphology and electrochemical behavior. *Acta Metallurgica Sinica, 49*, 43–50. https://doi.org/10.3724/SP.J.1037.2012.00361

Yang, Y., Wang, S., & Wen, C. (2016). Experimental study on alternating current corrosion of pipeline steel in alkaline environment. *International Journal of Electrochemical Science, 11*, 7150–7162. https://doi.org/10.20964/2016.08.64

Yang, Y., Walther, J. H., Yan, Y., & Wen, C. (2017). CFD modelling of condensation process of water vapor in supersonic flows. *Applied Thermal Engineering, 115*, 1357–1362. https://doi.org/10.1016/j.applthermaleng.2017.01.047

Yang, Y., & Wen, C. (2017). CFD modeling of particle behavior in supersonic flows with strong swirls for gas separation. *Separation and Purification Technology, 174*, 22–28. https://doi.org/10.1016/j.seppur.2016.10.002

Yoshida, H., Yoshikawa, S., Fukui, K., & Yamamoto, T. (2008). Effect of multi-inlet flow on particle classification performance of hydro-cyclones. *Powder Technology, 184*, 352–360. https://doi.org/10.1016/j.powtec.2007.09.008

Zhao, B., Su, Y., & Zhang, J. (2006). Simulation of gas flow pattern and separation efficiency in cyclone with conventional single and spiral double inlet configuration. *Chemical Engineering Research and Design, 84*, 1158–1165. https://doi.org/10.1205/cherd06040