CHEMICAL ENGINEERING TRANSACTIONS

VOL. 59, 2017

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.aidic.it/cet

Guest Editors: Zhuo Yang, Junjie Ba, Jing Pan

ISBN 978-88-95608- 49-5; **ISSN** 2283-9216

DOI: 10.3303/CET1759171

Numerical Investigation on the Effects of Pipe Rotation on a Dynamic Hydrocyclone

Songsheng Deng, Long Huang*, Jinfa Guan, Ming Chen

Department of Military Oil Supply Engineering, Logistic Engineering University of PLA, Chongqing 401331, China
hlong0000@163.com

Dynamic hydrocyclone and compound hydrocyclone are both new high efficiency separators whose difference is whether the pipe wall (wall of separation chamber) is rotating when working, but the influences of rotating pipe on hydrodynamics and performance of dynamic hydrocyclone has not been well studied. In this paper, a three dimensional numerical simulation applying Reynolds Stress Model (RSM) coupled with Multiple Reference Frame (MRF) model and algebraic slip mixture model was conducted to analyze flow field of dynamic hydrocyclone with and without pipe rotation. The results show that, pipe rotation can accelerate the flow around the wall leading to an increase of tangential velocity as well as avoidance of large velocity gradient around the wall. On the other hand, pipe rotation has little impact on axial velocity. And investigation on the turbulence shows that pipe rotation may effectively attenuate turbulent transport. What's more, pipe rotation significantly improves aggregation effect of oil droplet, which leads to an increase of separation performance.

1. Introduction

Known as a multiphase separator with high separation efficiency, small space requirement, and low energy consumption, hydrocyclone is gradually replacing the conventional vessel separator in the petroleum industry, and has been popularized to various fields (Papoulias and Lo, 2015). However, the swirling flow in a conventional hydrocyclone is established by a tangential inlet, so the swirl intensity is closely related to the inlet pressure or flow rate. Thus the conventional hydrocyclone is handicapped by some inherent limits such as the necessity to have a high pressure inlet to get sufficient acceleration field and the loss of efficiency at low flow rate. Considering these drawbacks of conventional hydrocyclones, dynamic hydrocyclone (DH) was developed by TOTAL CFP and NEYRTEC (Gay et al., 1987). The structure diagram of oil-water DH is shown in Figure 1, the main difference against static hydrocyclone is that fluid enters DH from an axial inlet, and is rotated by synchronous rotating blades and pipe (separation chamber wall) driven by a motor. Under the centrifugal force produced by swirling flow, lighter phase migrates towards the centerline then exits from overflow outlet and heavier phase moves towards the wall then exits from the underflow outlet. The performance of DH has been investigated by several researchers (Chen et al., 2015; Jones, 1993). the results showed that DH can remove smaller oil droplets from water and can operates efficiently at lower inlet pressures compared with conventional hydrocyclones.

In recent years, some researchers (Wang, et al., 2005) designed a new hydrocyclone called “compound hydrocyclone (CH)” which is a combination of dynamic and static hydrocyclones. It has a similar structure as DH with motor-driven blades to produce swirling flow, but its separation chamber is not rotating. The design intent of CH was to improve reliability of DH by diminishing a moving part — the rotating pipe wall. However, in the research of rotating pipe flow, it has been proved that rotating pipe had a significant effect on turbulence transfer, velocity distribution and friction loss (Imao et al., 1996). And the turbulence and velocity distributions are important factors affecting separation performance of hydrocyclone. But the specific influence of rotating pipe on DH remains unknown.

Computational Fluid Dynamics (CFD) is an useful tool in industrial and non-industrial applications. Compared with experiment method, CFD is a cheap and efficient way to provide valuable quantitative insight into the hydrodynamic behaviours of flow. It makes the industry more efficient and productive and have become very

Please cite this article as: Songsheng Deng, Long Huang, Jinfa Guan, Ming Chen, 2017, Numerical investigation on the effects of pipe rotation on a dynamic hydrocyclone, Chemical Engineering Transactions, 59, 1021-1026 DOI:10.3303/CET1759171

popular for simulating flow fields and performance in many areas (Guo et al., 2016; Song et al., 2016). In this paper, the flow fields of DH with and without pipe rotation are studied by CFD method whose validity and accuracy is verified by experiment data. The effects of rotating pipe on velocity profile, turbulence distribution and oil-water separation performance are deeply investigated. The results of this paper shed light on flow and separation mechanism in DH and are favour of its design.

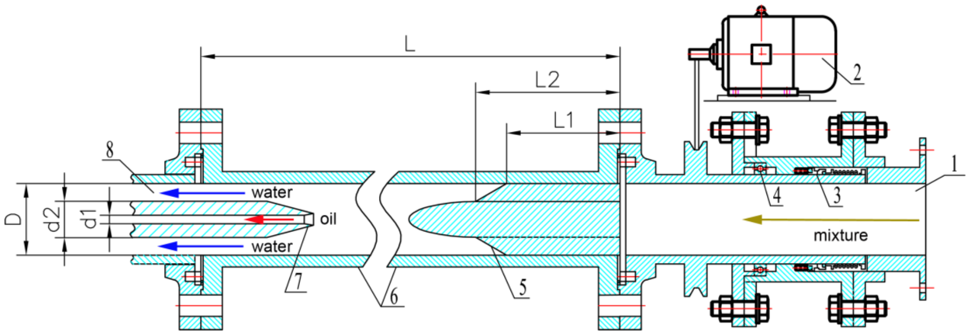


Figure 1: Diagram of dynamic hydrocyclone. 1 - Inlet; 2 - Motor; 3 - Mechanical seal; 4 - Bearing; 5 - Blades; 6 - Rotating pipe; 7 - Overflow outlet; 8 - Underflow outlet

2. Model description

2.1 Turbulence model

The most widely used turbulent models for cyclone separators are κ-ε model and the Reynolds stress model (RSM). The RSM solves Reynolds stress transport equations directly, and thus avoids the use of an eddy viscosity and is well suited for handling the anisotropic turbulence fluctuations (Liu and Gao, 2015). The RSM always provides more accurate results than model, so RSM was adopted in this study.

2.2 Moving zone model

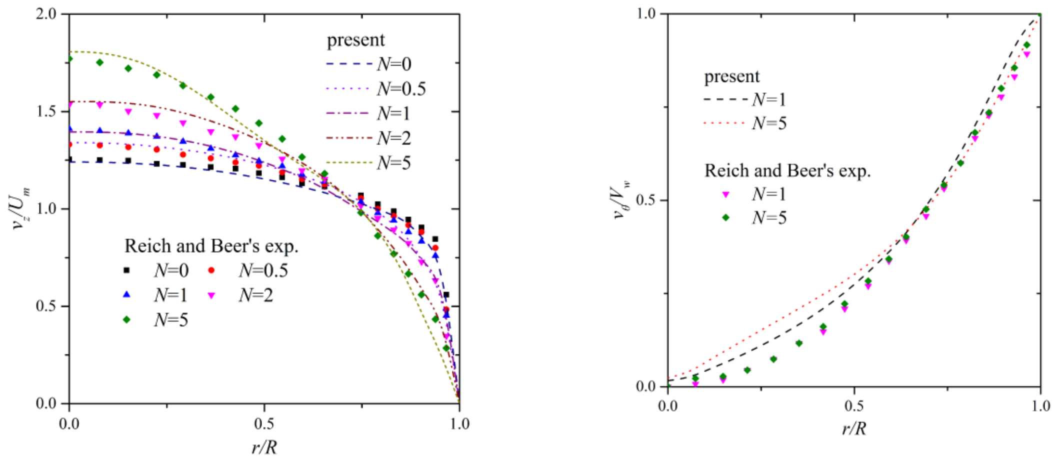
The blades and separation chamber wall of DH rotate all the time when working, so the simulation involves moving zone. There are 3 models for mobile zone problems, Multiple Reference Frame (MRF), Sliding Mesh (SM) and Dynamic Mesh (DM). MRF is a steady-state approximation in which mesh doesn’t really move, and the equations of moving zone are calculated in a moving coordinate system. While In SM and DM methods, the mesh actually moves. And these two methods calculate the transient flow field, and cost considerably more than the MRF method. Under stable working condition, the flow in DH can be seen as steady flow, and it has been proved that the MRF model can provide a reasonable result of the time-averaged flow for many applications (Deglon and Meyer, 2006; Yang and Zhou, 2015). So MRF is suitable for the simulation.

2.3 Multiphase model

To simulate oil-water two-phase flow, algebraic slip mixture model is applied in this work. It is a good substitute for the full Eulerian multiphase model especially when the entrance volume fraction of dispersed phase is lower than 10% (Paladino et al., 2005). The mixture model can model several phases (fluid or particulate) by solving the momentum, continuity, and energy equations for the mixture, the volume fraction equations for the secondary phases, and algebraic expressions for the relative velocities. It’s suitable for the simulation of cyclone separators.

2.4 Model validation

In order to verify the feasibility of the simulation method, flow field in a rotating pipe in Reich and Beer’s experiment (1989) was first simulated by the same method in this work. The simulation results and experimental data of Reich and Beer have been plotted in Figure 2. It can be seen from Figure 2 that the flow field was predicted with high accuracy using the simulation method described above. Figure 2(a) shows that axial velocity profiles of present work have little distinct with Reich and Beer’s experimental data, and the laminarization phenomenon is well presented when *N* increase. Tangential velocities are compared in Figure 2(b) and also show good agreement, calculated tangential velocity grows with radial distance in a quadratic function, and shares the similar tendency with experimental data. So it can be concluded that the models mentioned above are feasible for the simulation of swirling flow in rotating pipe.



(a) Axial velocity *(b) Tangential velocity*

Figure 2: Comparison of velocity distribution between experimental data (Reich and Beer, 1989) and results of present simulation; v_z– axial velocity, U_m– axial mean velocity, v_θ– tangential velocity, V_w– tangential velocity at the pipe wall, rotation rate N= V_w/U_m

3. Computational details

3.1 Computational domain and grid arrangement

As shown in Figure 1, the diameter (D) of DH studied in this work is 50 mm; the length of separation chamber (L) is 800 mm; the lengths of blades (L1 and L2) are 85 mm and 100 mm respectively; the diameter of overflow outlet (d1) is 10 mm and the inner diameter of annular underflow outlet (d2) is 30 mm. Due to the complicated structure, it's quite challenging to generate a good grid that can accurately predict the studied flow field. With repeated attempts, structure grid was successfully divided by software ICEM. Figure 3 shows an overall view of the generated grid. The total number of cells is 240,143, and the grid in the vicinity of the inlet, outlet and blades was refined.

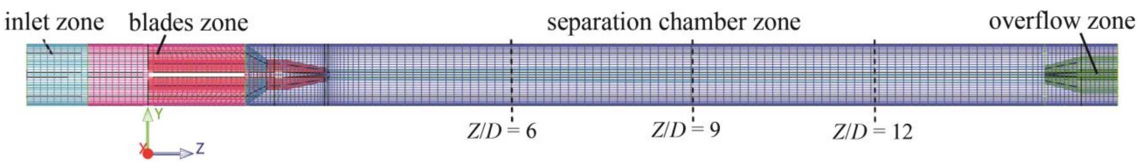


Figure 3: Grid and boundary conditions

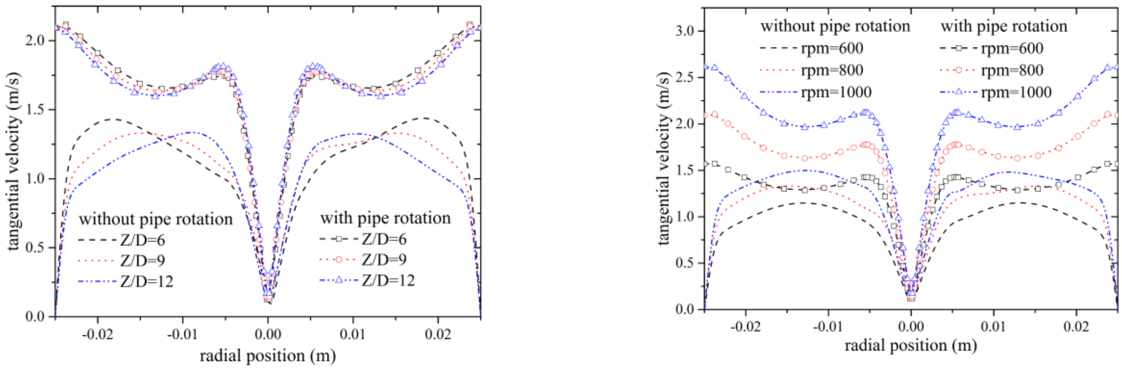
3.2 Boundary conditions

The boundary conditions are also illustrated in Figure 3. Flow enter the hydrocyclone though "velocity inlet" where the value of velocity were calculated from the flow rate which is 3 m3/h. Overflow and underflow outlet boundaries were set as "outflow", with the flow split ratio of 10%. The oil referred herein is diesel with density of 810 kg/m3 and viscosity of 0.004 Pa•s. The density of water is 1,000 kg/m3, and the viscosity is 0.001 Pa•s. In order to investigate the influence of rotating pipe, computational domain was divided into several zones: the inlet zone and overflow zone were stationary; blades zone was rotating; and separation chamber zone was set stationary or rotating in different cases, and the different zones were connected by interfaces. Rotation speed of 600, 800 and 1,000 rpm were studied

4. Results and discussions

4.1 Boundary conditions

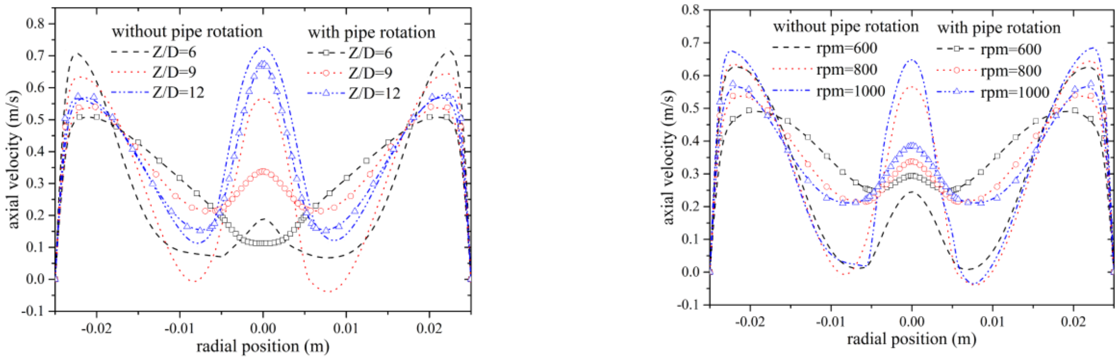
Tangential velocity is the most important velocity component in hydrocyclone, because the centrifugal force that separates oil and water relates directly to tangential velocity. Figure 4 illustrates tangential velocity for both cases with and without pipe rotation. When only the blades rotate, tangential velocity profiles are similar to static hydrocyclone. The flow behaves like Rankine vortex, in which the flow is of the "forced vortex" type in the center and of the "free vortex" type in outer space. In the center, tangential velocity is proportional to the distance to pipe centerline. And in outer space, tangential velocity decreases slowly from extreme point to the wall, and it should be mentioned that there exists large velocity gradient near the wall which may lead to droplet breakage. However, when blades and pipe wall rotate synchronously, it is found that the flow also can be divided into inner and outer parts. The inner flow is still “forced vortex”, but in outer flow the tangential velocity no longer decreases but increases along radial direction. It can be inferred that rotating pipe accelerates the flow near the wall, making the tangential velocity in outer flow remains high. Another effect of rotating pipe can be seen from Figure 4(a), tangential velocity becomes smaller when the axial distance increase in the case without pipe rotation, but it decays little with increase of axial distance in the case with pipe rotation. It indicates that rotation is slowed down by friction against the wall of cyclone. Larger tangential velocity can provides larger centrifugal force and it’s beneficial for separation.



(a) (b)

Figure 4: Comparison of tangential velocity distribution in different cases. (a) different cross sections at rotation speed of 800rpm; (b) different rotation speed at the same cross section Z/D=9

Figure 5 indicates the axial velocity distribution of DH in different cases. From the centreline to the wall, axial velocity decreases at first, then increases and finally decreases again. And it is found, through comparison, that pipe rotation has not much influence on axial velocity, curves of two cases show the same trend. The only difference is that pipe rotation makes the curves more flat. What’s more, from Figure 5(a), it can be seen that axial velocity becomes larger in the center and smaller around the wall when the cross section gets closer to outlet. Figure 5(b) shows a trend that increasing rotation speed may cause axial velocity curve to become steeper.



(a) (b)

Figure 5: Comparison of axial velocity distribution in different cases. (a) different cross sections at rotation speed of 800rpm; (b) different rotation speed at the same cross section Z/D=9

4.2 Turbulence

In Figure 6, turbulent intensity is shown for both cases at section Z/D=9. Turbulent intensity is defined as the ratio of the root-mean-square of the velocity fluctuations to the mean flow velocity, and it reflects strength of turbulence. As is seen, rotating pipe can effectively decrease turbulent intensity, especially around the wall. With pipe rotation, turbulent intensity is below 7%, while without pipe rotation, it’s larger than 10% when rotation speed is high. What's more, when rotation speed increase, without pipe rotation, turbulent intensity grows sharply; with pipe rotation, turbulent intensity increases a little. The results agree with findings in the research of rotating pipe flow that pipe rotation leads to attenuation of turbulent transport (Imao et al., 1996). This phenomenon is beneficial for separation, because turbulence is one of the causes that limit the performance capabilities of hydrocyclone (Gay et al., 1987).

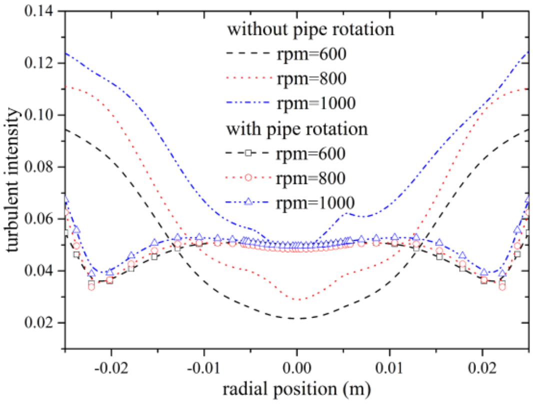


Figure 6: Comparison of turbulent intensity distribution with and without pipe rotation

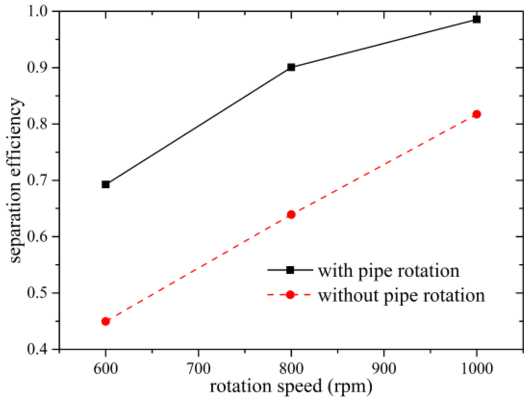


Figure 7: Comparison of separation efficiency

4.3 Oil-water separation performance

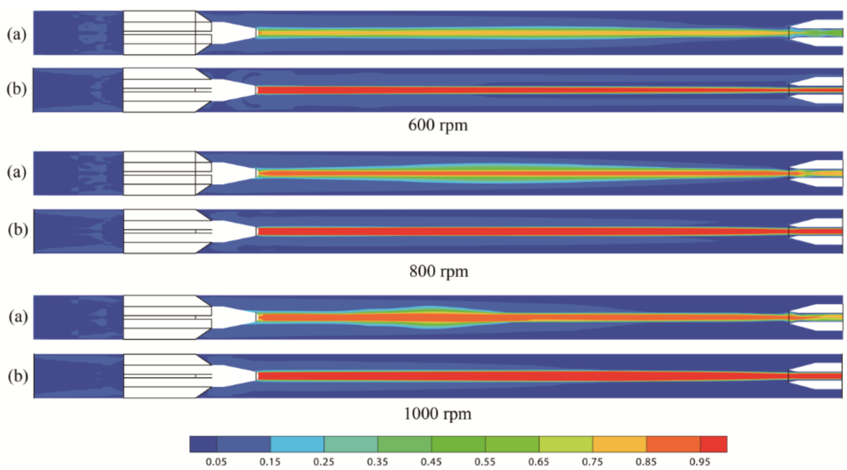


Figure 8: Contours of phase friction of oil, (a) without pipe rotation; (b) with pipe rotation

Figure 8 shows the contours of phase friction of oil (diesel) in DH. As is seen, thanks to the axisymmetric structure and axial inlet, the oil phase distribution is also axisymmetric. Under centrifugal force, oil is aggregated towards center, and the aggregation effect gets better when rotation speed grows. Comparing the cases with and without pipe rotation, it could be seen that pipe rotation can improve the aggregation effect. With pipe rotation, the oil fraction is larger in the center and smaller in the outer space. Improvement of aggregation of oil leads to an increase of separation efficiency, as shown in Figure 7. Obviously, pipe rotation improves separation efficiency largely, the improvement is about 20%. So, theoretically, it's better to have the pipe rotate together with the blades to gain better separation performance for dynamic hydrocyclone.

5. Conclusions

CFD approach based on finite volume method was used to investigate the effect of rotating pipe on a DH. RSM coupled with MRF model were utilized to simulate the dynamic turbulent flow, and algebraic slip mixture model was applied to simulate oil-water two-phase flow. The numerical method was validated by experiment of rotating pipe flow. The flow fields of DH with and without pipe rotation were compared, and the results indicate that pipe rotation has significant influence on velocity and turbulence distributions. With pipe rotation, the tangential velocity is accelerated; turbulence intensity is reduced and efficiency is promoted. From this point, it's better for DH to run with a rotating pipe. However, the rotating pipe is exposed to air and the mechanical reliability needs to be test further more.

Acknowledgments

The authors acknowledge the financial support of Chongqing Research Program of Basic Research and Frontier Technology (Grant No. cstc2016jcyjA0171).

Reference

Chen J.W., Hou J., Li G., Xu C., Zheng B., 2015, The Effect of Pressure Parameters of a Novel Dynamic Hydrocyclone on the Separation Efficiency and Split Ratio, Separation Science and Technology, 506, 781-787.

Deglon D.A., Meyer C.J., 2006, CFD modelling of stirred tanks: Numerical considerations, Minerals Engineering, 1910, 1059-1068.

Gay J.C., Triponey G., Bezard C., Schummer P., 1987, Rotary cyclone will improve oily water treatment and reduce space requirement/weight on offshore platforms, In Offshore Europe. Society of Petroleum Engineers, Aberdeen, UK.

Guo Q.J., Qi X.N., Wei Z., Guo P.J., Sun P., 2016, 3D numerical simulation and analysis of refrigeration performance of the small diameter vortex tube, International Journal of Heat and Technology, 34(3), 513-520. DOI: 10.18280/ijht.340323

Imao S., Itoh M., Harada, T., 1996, Turbulent characteristics of the flow in an axially rotating pipe, International Journal of Heat and Fluid Flow, 17, 444-451.

Jones, P.S., 1993, A field comparison of static and dynamic hydrocyclones, SPE Production & Facilities, 2, 84-90.

Li H., Gao, B., Li B., 2015, Numerical Analysis of Flow Dynamics of Cyclone Separator Used for Circulating Fluidized Bed Boiler. Chemical Engineering Transactions, 46, 991–996. DOI:10.3303/CET1546166

Paladino E.E., Nunes G.C., Schwenk L., 2005, CFD analysis of the transient flow in a low-oil concentration hydrocyclone, In Conference Proceedings of the AIChE Annual Meeting, pp. 646-657.

Papoulias D., Lo, S., 2015, Advances in CFD Modelling of Multiphase Flows in Cyclone Separators. Chemical Engineering Transactions, 43, 1603–1608.

Reich G., Beer H., 1989, Fluid flow and heat transfer in an axially rotating pipe—I. effect of rotation on turbulent pipe flow, International Journal of Heat & Mass Transfer, 32, 551-562.

Song H.J., Zhang W., Li Y.Q., Yang Z.Y., Ming A.B., 2016, Simulation of the vapor-liquid two-phase flow of evaporation and condensation, International Journal of Heat and Technology, 34(4), 663-670. DOI: 10.18280/ijht.340402

Speziale C.G., Younis B.A., Berger S.A., 2000, Analysis and modelling of turbulent flow in an axially rotating pipe, Journal of Fluid Mechanics, 407, 1-26.

Wang Z.C., Yu H.B., Liu X.M., 2005, Research on the design of a de-oiling compound hydrocyclone and its separation characteristics, Petroleum Science, 3, 28-32.