CHEMICAL ENGINEERING TRANSACTIONS

VOL. 46, 2015

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

Guest Editors: Peiyu Ren, Yancang Li, Huiping Song

ISBN 978-88-95608-37-2; **ISSN** 2283-9216

DOI: 10.3303/CET1546166

Numerical Analysis of Flow Dynamics of Cyclone Separator Used for Circulating Fluidized Bed Boiler

Haixia Li[a], Bingguang Gao[b], Bin li[a]

[a] School of Mechanical and Power Engineering, Henan Polytechnic University, Jiaozuo 454000, China,
[b] School of Application Techniques, Henan Polytechnic University, Jiaozuo 454000, China.
lihx@hpu.edu.cn

The Reynolds stress model (RSM) provided by FLUENT software was applied to study flow dynamics of a cyclone separator used for a circulating fluidized bed. The effect of the depth of the exhaust pipe on the flow field and performance of cyclone separator was investigated computationally. The cyclone flow field pattern has been simulated and analyzed with the aid of velocity components and static pressure distribution. The simulation results reveal the strong swirling flow in cyclone separator. The exhaust pipe depth has an insignificant effect on the pressure drop and velocity distribution.
Keywords: cyclone separator; Reynolds stress model (RSM); numerical simulation; exhaust pipe

1. Introduction

Gas cyclone separators are widely used in industries to separate dust from gas due to their geometrical simplicity and relative economy in power consumption (Elsayed and Lacor, 2011). Cyclones may also be adapted for use in extreme operating conditions such as high temperature, high pressure, and corrosive gases. Since there are no moving parts. cyclones are relatively maintenance-free. Therefore, cyclones have found increasing utility in the field of air pollution, the power plant and process industries.
The cyclone separator plays a major role in circulating fluidized bed boiler operation in power plant. A large number of solid materials were separated from the flue gas, and then taken back into the furnace chamber, establishing furnace material circulation (Zhao (2010)). This can ensure the fuel and desulfurizer in cycling combustion reaction repeatedly for many times. So, the circulating fluidized bed boiler has good combustion and desulfurization efficiency and the circulating fluidized bed boiler can run in full load. The separation performance and operation status of cyclone separator has direct relationship with the combustion, desulfurization and circulating rate of circulating fluidized bed boiler (Hui et al. 2008).
Until now, a considerable number of investigations has been performed on small sampling cyclones (Souza et al. 2015, Safikhani et al. 2010, Chen et al. 2010, Lu et al. 2015 and Bhasker, 2010).The investigation of full size cyclone separator was seldom reported in the literatures. In addition, the exhaust pipe depth displays a great effect on the flow conditions and performance of the cyclone separator during process of use in power plant. However, the analysis of the effect of the exhaust pipe depth on cyclone separator is very rare. Therefore, the effect of the exhaust pipe depth on the cyclone separator used for a certain 240t/h circulating fluidized bed boiler of a power plant was investigated in this paper is. The numerical simulation method is applied to simulate the flow field of full size cyclone separator. The research results can provide guidance for industrial application of cyclone separator.

2. Models and boundary conditions

2.1 Computational domain and domain discretion

Fig. 1 shows the schematic of the cyclone separator which is applied to some 240t/h circulating fluidized bed boiler. The size of the cyclone separator is the same as the size in industry without any simplification. The cyclone separator consists of inlet segment, straight cylinder segment, conical barrel segment, exhaust pipe and ash discharging pipe.

Please cite this article as: Li H.X., Gao B.G., 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

Fig. 2 shows the grid of cyclone separator. The hexahedral grid is used to the whole calculation region according to the structure characteristics of the cyclone separator to improve the calculation accuracy and to control grid number effectively. Several sets of grids were used to determine if there are sufficient grid points. There is no obvious difference in pressure distribution when the grid cells are further increased. Therefore, the grid set is independent grid that ensured sufficient accuracy of the simulation results. The total grid number is 320000 in this model.

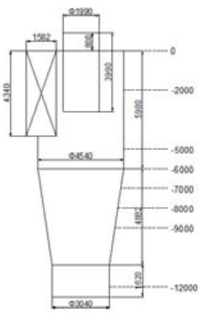


Figure 1: Schematic diagram of cyclone structure.

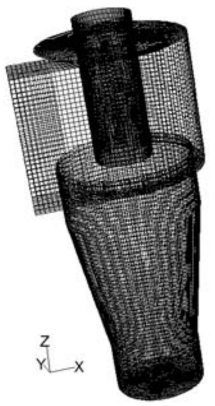


Figure 2: Schematic diagrams of grids of filter vessel.

2.2 Simulation model and boundary conditions

The FLUENT code has provided the k-ε and the Reynolds stress transport model (RSTM) and other turbulent models for the users to choose. The RSTM accounts for the evolutions of the individual stress components (Wang et al. (2010), Liu et al. (2005) and Yu et al. (2002)), and thus entirely avoids the use of an eddy viscosity and is well suited for handling the anisotropic turbulence fluctuations (Oh et al. (2015)). So, in this work, it is applied to simulate the gas flow in view of the flow condition in cyclone separator. The volume averaged conservation equation in the cyclone separator is discretized by the control volume finite element method (Khedher (2014) and Bounaouara (2015)). The pressure drop of the cyclone separator obtained from the numerical simulation agrees well with the pressure drop measured by experiment in power plant. So it can be concluded that the simulation model suggested in this study is useful to predict the flow dynamics of the cyclone separator. The upwind 2nd order discretization scheme was applied when the governing equations were iterated, till the maximum target convergence value of order .0001 was reached.

The gas inlet boundary is set as velocity boundary condition with velocity of 15m/s. The values for turbulence intensity and length scale are critical and carefully assigned for inlet boundary conditions. The full development boundary condition is applied for the gas outlet boundary condition. The wall boundary condition is no slip boundary condition.

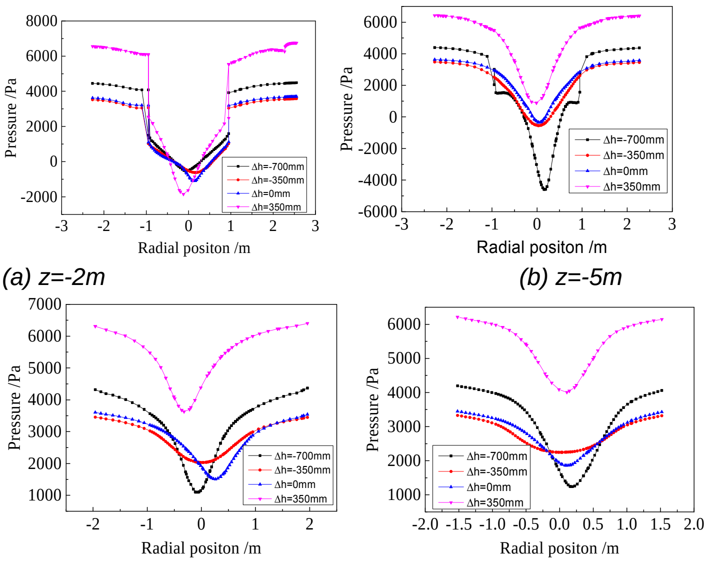
3. Numerical simulation results and analysis

3.1 Pressure distribution

The flow field was studied by analyzing the pressure and velocity distribution in different height section in cyclone separator. z=-2m is located in circular region. z=-5m is located in straight cylinder region. z=-8m is located in conical barrel region. z=-12m is located in ash discharging region.

The depth of the exhaust pipe was changed to investigate the effect of its length on the flow characteristics in cyclone separator. The distance between the bottom of the exhaust pipe and the bottom of the inlet of the cyclone separator were -700mm, 0mm and 350mm, respectively. The minus means that the bottom of the exhaust pipe is relative lower than the bottom of the inlet of the cyclone separator. For example, -700 means that the bottom of the exhaust pipe is 700mm lower than the bottom of the inlet of cyclone separator. 0mm means that the bottom of the exhaust pipe and the cyclone inlet are at the same horizontal plane.

Fig. 3 presents the pressure variation with the depth of the exhaust pipe alone radial direction at different heights section. It can be seen that the static pressure is higher near cyclone separator wall than the pressure in the central part of the cyclone separator. The internal swirl flow appears in central region of the cyclone separator. Therefore, the lowest pressure occurs in this region. The static pressure distribution is essentially symmetric in radial position. The deviation from symmetric part is mainly due to the cyclone inlet airflow which enters the cyclone separator tangentially. The static pressure decreases from the ash discharging tube to the gas exhaust pipe entrance. The pressure in the central position of the inlet segment decreases with the decrease of the depth of exhaust pipe. The pressure difference across the surface of the exhaust pipe is the largest at the case of Δh=350mm. In this case, the exhaust pipe should suffer the most large pressure force.

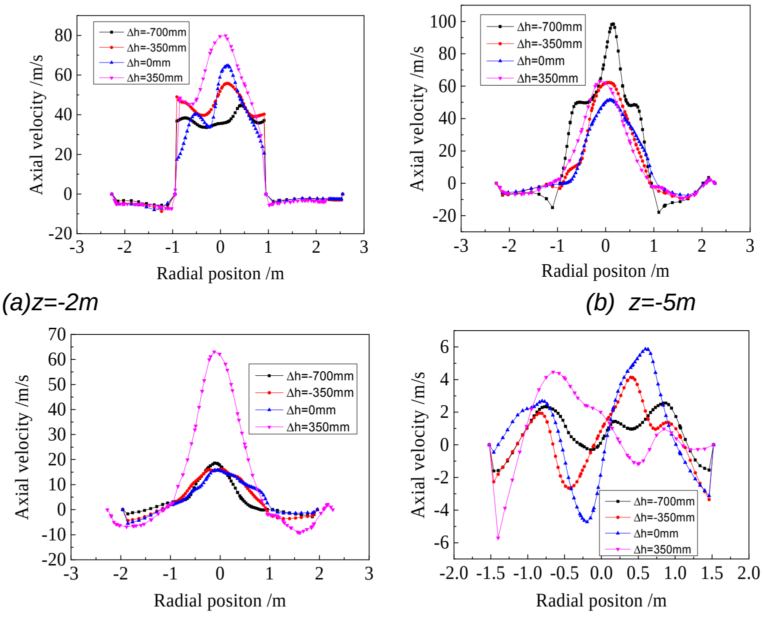


(a) z=-2m *(b) z=-5m* *(c) z=-8m* *(d) z=-12m*

Figure 3: The distribution of pressure at different insertion depth of exhaust pipe.

3.2 Velocity distribution

Fig. 4 shows axial velocity varying with the depth of exhaust pipe along radial position at different height section. The axial velocity distribution is essentially symmetrical in the straight cylinder and conical barrel segment (Song et al. (2005)). Gas flows downwards near cyclone separator wall while gas flows upwards in the central of cyclone separator. The axial velocity in central part of the inlet segment increases as the depth of exhaust pipe decreases. The axial velocity distribution at Δh=350mm will lead to more little particle escape from exhaust pipe. The axial velocity is almost the same at annular separation space. The maximum axial velocity does not occur at the center part of the ash bucket, therefore, there is no obvious retention of particle in this section.

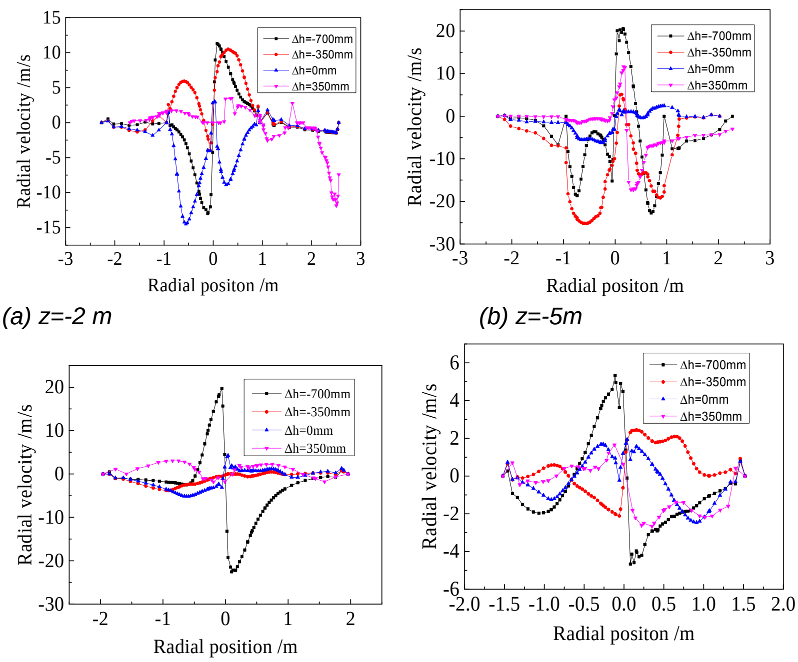


(a)z=-2m (b) z=-5m

(c)z=-8m (d) z=-12m

Figure 4: The distribution of axial velocity of different insertion depth

Fig. 5 shows radial velocity variation along radial position at different heights. The radial velocity decreases with the radius increasing. The distribution trend of radial velocity changes little at different heights. An identical flow phenomenon was observed at the cyclone. The radial velocity profiles show that the flow in cyclone separator is rotational and approximately symmetrical, which enhances the transport of particles onto the cyclone walls.

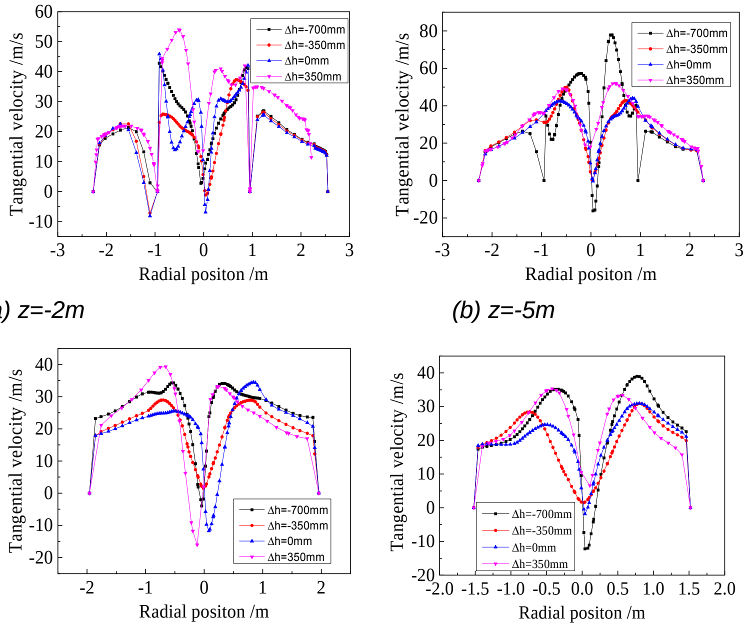


(a) z=-2 m (b) z=-5m

(c) z=-8m (d) z=-12m

Figure 5: The distribution of radial velocity of different insertion depth

The tangential velocity profiles are shown in Fig. 6 at different cross sections for different exhaust pipe depth. The tangential velocity increases with the increase of radius in the central of cyclone separator. In this region, the fluid has the same angular velocity at different radial location. The swirling flow in this region is called forced vortex flow. The tangential velocity decreases with the increases of the radius at the region outside the central of cyclone separator. In this region, the fluid momentum is in conservation and the flow is described as free vortex flow. The depth of the exhaust influences the tangential velocity distribution more in the place of the inlet segment and surrounding the inlet of the exhaust pipe.



(a) z=-2m *(b) z=-5m*

(c) z=-8m *(d) z=-12m*

Figure 6: Tangential velocity variation with radial position at different insertion depth.

4. Conclusions

The Reynolds stress model (RSM) provided by FLUENT software was applied to study flow field of a cyclone separator used in some circulating fluidized bed.

The static pressure distribution is essentially symmetric in radial position. The static pressure decreases from the ash discharging tube to the gas exhaust pipe entrance.

Gas flows downwards near cyclone separator wall while gas flows upwards in the central of cyclone separator. The swirling flow in this central region is forced vortex flow and the flow is free vortex flow.

The depth of exhaust pipe influences the pressure distribution and velocity distribution near the bottom of the exhaust pipe. The pressure difference will increase when the bottom of the exhaust pipe is located at the top of the inlet of cyclone separator. The axial velocity is higher at the central part of exhaust pipe in the inlet segment when the exhaust pipe inlet is upper than the inlet bottom of the cyclone separator.

Acknowledgments

This work was financially supported by the national natural science foundation of china (No. U1504217), Technological Major Special Project of Henan Province in China (No.508057), Scientific and technological project of Henan Province(102102210209) and Natural Science Foundation of Henan Province (2010B470005).

References

Bhasker C., 2010, Flow simulation in industrial cyclone separator, Advances in Engineering Software, 41, 220-228.

Bounaouara H., Ettouati H., Ticha H.B., Mhimid A., Sautet J.C., 2015, Numerical simulation of gas-particles two phase flow in pipe of complex geometry: Pneumatic conveying of olive cake particles toward a dust burner, International Journal of Heat and Technology, 33(1), 99-106.

Chen J., Liu X., 2010, Simulation of a modified cyclone separator with a novel exhaust, Separation and Purification Technology, 73,100-105.

Duan L., Wu X., Ji Z., Fang Q., 2015, Entropy generation analysis on cyclone separators with different exit pipe diameters and inlet dimensions, Chemical Engineering Science, 138,622–633.

Elsayed K., Lacor C., 2011, Numerical modeling of the flow field and performance in cyclones of different cone-tip diameters, Computers & Fluids, 51, 48–59.

Hui S., Ji G., Jin Y.,Wang J., 2008, Investigation on separation performance of cyclone separator used in circulating fluidized bed boiler, China Powder Science and Technology, 14(2),42-44, 10.3969/j.issn.1008-5548.2008.02.013.

Khedher N. , 2014, Three-dimensional simulation of the thermal performance of porous building brick impregnated with phase change material,International Journal of Heat and Technology, 32 (1),163-169.

Liu S., Zhang Y., Wang B.,2005, Cyclone Separator Three-Dimensional Turbulent Flow-Field Simulation Using the Reynolds Stress Model, Transactions of Beijing Institute of Technology,25(5),377-380, 10.3969/j.issn.1001-0645.2005.05.001.

Mao Y., Pang L., Wang X., Wu D., 2002, Numerical modeling of three dimension turbulent field in cyclone separator, Petroleum Processing and petrochemicals, 32(2), 32-37, 10.3969/j.issn.1005-2399.2002.02.001.

Oh J., Choi S., Kim J., 2015, Numerical simulation of an internal flow field in a uniflow cyclone separator, Powder Technology, 274, 135-145.

Safikhani H., Akhavan-Behabadi M.A., Shams M., Rahimyan M.H., 2010, Numerical simulation of flow field in three types of standard cyclone separators, Advanced Powder Technology, 21, 435-442.

Song J., Wei Y., Shi M., 2005, Asymmetry of gas-phase flow field in cyclone separator. Journal of Chemical Industry and Engineering, 56(8), 55-60, 10.3321/j.issn:0438-1157.2005.08.004.

Souza F., Salvo R., Martins D., 2015, Effects of the gas outlet duct length and shape on the performance of cyclone separators, Separation and Purification Technology, 142, 90-100.

Wang J., Mao Y., Liu M., Wang J., 2010, Numerical simulation of strongly swirling flow in cyclone separator by using an advanced rng k-εmodel, Acta Petrolei Sinica, 26(1),8-13, 10.3969/j.issn.1001-8719.2010.01.002.

Zhao L., 2010, Center tube renewal analysis of 240 t/h circulating fluidized bed boiler, Plant Maintenance Engineering, 8, 41-42, 10.3969/j.issn.1001-0599.2010.08.023.