Pressure Drop Determination in a Novel Cyclon Separator by Using Numerical Modeling Technique

Roman Havryliv, Vladyslav Bodnaruk, Vitalii Trynchuk

Department of chemical engineering, Lviv Polytechnic National University, UKRAINE, Lviv, St. Yura square, 9, E-mail: havrilivroman@gmail.com

Abstract – This study presents numerical simulation results of the pressure drop determination in a novel cyclone design separator. ANSYS Fluent 16 software was used to numerical modelling. The Reynolds-averaged Navier–Stokes equations with the k-ε RNG, k-ε Realizable and Reynolds Stress turbulence Models (RSM) were used in the analysis.

Keywords – cyclone separator, pressure drop, numerical modeling, turbulence model.

Introduction

Cyclones as centrifugal separation apparatus have been used in chemical engineering for many decades. It's a very simple construction device with high separation efficiency. The recent year's tendencies to reduce energy costs in the industrial processes require the creation of the new cyclone separators design such as cyclones with the co-flow work zone [1,2].

Because experimental studies in industrial conditions are very expensive as a powerful tool for assessing the effectiveness of created cyclones, CFD (Computational Fluids Dynamic) technique can be used. In the present study, ANSYS Fluent 16 software was used to numerical simulation and the pressure drop determination in a novel cyclone design separator.

Experimental Setup and Modeling Results

The pressure drop and separation performance strongly depend on the structure and length of the gas swirling flow formed in the cyclone. These are the main technical parameters that characterize the perfection of the any cyclones design and are affected mainly by the cyclone geometry. Analysis of velocity fields with different shapes and configurations of the various cyclones design shows that use of a direct-flow zone to improve the separation performance and pressure drop reduces, will be effective.

The modeled novel cyclone with the co-flow work zone is shown in Fig. 1 and the structural dimensions are given in Table 1. The computational 3D domain of the cyclone geometry was discretized with unstructured mesh. The resulting mesh contains about 855000 Nodes and 4400000 Elements. All geometric transformations were done in a software module Design Modeller, and the mesh was created in ANSYS Meshing.

To simulate the swirling flow inside the cyclone the basic equations of hydrodynamics, namely the continuity Eq.1 and the equation of momentum conservation Eq.2 were used:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \vec{u}) = S_m \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \vec{u}) + \nabla(\rho \vec{u}\vec{u}) = -\nabla p + \nabla(\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \quad (2)$$

where ρ − density, $\vec{u}$ − overall velocity vector, S_m − the mass added to the continuous phase from the dispersed second phase or any user-defined sources, p − static pressure, $\bar{\bar{\tau}}$ − stress tensor, ρg − the gravitational body force, F − external body force.

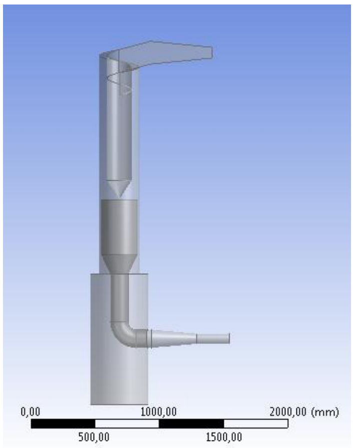

Fig.1. Geometry of cyclone.

In addition to the basic model equations of the medium flow, several turbulent models are available in Ansys Fluent software. The k-ε RNG, k-ε Realizable and Reynolds Stress turbulence Models (RSM) were used in the modeling. The system of differential equations was supplemented by relevant boundary conditions for the inlet, wall and outlet. Operation modes of the cyclone for inlet flow velocity was changes from 15 to 25 m/s.

Table 1

Dimensions of the cyclone tested in this study

Cyclone dimensions	Size, mm
Cyclone diameter	310
Work zone height	980
Central pipe height	960
Inlet pipe	175x60
Bunker diameter	450
Bunker height	1000
Coax insertion pipe diametr	286

The simulation results were compared with the measured pressure drop experimental data. It's was found that the RSM simulation results match the experimental pressure drop profile. The relative error between experiment and CFD was less than 10%.

Conclusion

The pressure drop results of movements swirling turbulent flow inside the cyclone separator had great deviation when using RNG and Realizable k–ε models. But the prediction results by RSM well consistent with the experimental value. In addition, a novel cyclone design to have a lower pressure drop (800 Pa) than the standard cyclone CN-15 (1200 Pa).

References

[1] R. I. Havryliv, V. V. Maystruk, O. O. Timura, O. M. Kuhar. “Determination of hydraulic resistance of cyclones by means of computer modeling Solidworks flow simulation”. *Scientific herald of the National Forestry University of Ukraine, Collection of scientific works,* vol. 23 (16), pp. 364 – 369, 2013.

[2] V. V. Maystruk, R. I. Havryliv, A. S. Popil, A. M. Basistyi. “Assessment of energy costs in work of direct-acting cyclone using software package flow simulation”. *Eastern-European Journal of Enterprise Technologies,* vol. 68 (60), pp.28- 31, 2012.