Journal of Basic and Applied Engineering Research
Print ISSN: 2350-0077; Online ISSN: 2350-0255; Volume 2, Number 1; January-March, 2015, pp. 67-68

http://www.krishisanskriti.org/jbaer.html

Computational Fluid Dynamics analysis for the Simulation of Cyclone Separator using CFX

Talat Fatima[1] and Syed Akhlaq Ahmad[2]

[1,2]Aligarh Muslim University, Aligarh, U.P.
E-mail: [1]talatfatima@yahoo.in, [2]sa.ahmad.ke@amu.ac.in

Abstract—*In this work the Reynolds stress model is applied for the simulation of cyclone separator using software CFX. Contours of pressure and axial velocity at transverse and longitudinal midsection of the cyclone have been studied. Streamlines of some randomly selected particles leaving the two outlets have also been shown.*

1. INTRODUCTION

Gas cyclone separators are widely used in industries to separate dust from gas streams or for product recovery because of its geometrical simplicity, relative economy in power consumption and flexibility with respect to high temperature and pressure. The conventional method of predicting the flow field and the collection efficiency of cyclone separator is empirical. During the past decades, application of computational fluid dynamics (CFD) for the numerical calculation of the gas flow field in a cyclone has become more popular. Earlier studies on CFD simulations were carried out using the standard *k*- ε turbulence model were found to be inadequate to simulate flows with swirl because it leads to excessive turbulence viscosities and unrealistic tangential velocities. Recent studies suggest that the accuracy of numerical solution can be improved by using Reynolds Stress Model (RSM).

2. MODELING EQUATIONS

Reynolds stress equation may be written in its most recognizable form as

$$\frac{\partial \tau_{ij}}{\partial t} + U_k \frac{\partial \tau_{ij}}{\partial x_k} = -\tau_{ij}\frac{\partial U_i}{\partial x_k} - \tau_{ik}\frac{\partial U_j}{\partial x_k} + \in_{ij} - \pi_{ij} + \frac{\partial}{\partial x_k}\left(\upsilon \frac{\partial \tau_{ij}}{\partial t} + C_{ijk}\right) \quad (1)$$

where $$\grave{o}_{ij} = 2\mu \frac{\partial u'_i}{\partial x_k}\frac{\partial u'_j}{\partial x_k} \quad (2)$$

$$\pi_{ij} = p'\left[\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i}\right] \quad (3)$$

$$C_{ijk} = \left[\left(\rho u'_i u'_j u'_k\right) + \left(p' u'_i\right)\delta_{jk} + \left(p' u'_j\right)\delta_{ik}\right] \quad (4)$$

Drag on the particle as given by Schiller Naumann Drag Model is modified in CFX to ensure the correct limiting behavior in the inertial regime and is given as

$$c_D = \max\left[\frac{24}{R_e}\left(1 + 0.15 R_e^{0.687}\right), 0.44\right] \quad (5)$$

The above model has been simulated using the CFX software.

3. RESULTS AND DISCUSSIONS

To study the contours of pressure and velocity of cyclone separator, a geometry with the dimensions as given in Table 1 was prepared in CFX modeller. A grid independency test was performed to select the mesh having 41744 elements. The geometry of the cyclone and grid presentation is shown is the Fig.1.

Table 1: Dimensions of Cyclone Separator (D = 0.2 m).

a/D	b/D	De/D	S/D	h/D	H/D	B/D
0.5	0.25	0.5	0.625	2	4	0.25

4. CONTOURS

Fig. 2 shows contours of pressure at transverse and longitudinal midsection of the cyclone. It can be seen that that the pressure is maximum at the boundary of the cyclone and goes on decreasing as we move towards the centre. At the centre the pressure is minimum which results in the formation of vortices that are required for the separation of heavier particles from the bottom outlet.

Fig. 3 shows the contour of axial velocity at transverse and longitudinal midsection of the cyclone. The axial velocity is maximum near the centre of the cyclone shown by red and yellow region. The positive magnitude shows that the velocity is upwards. At the bottom of the cyclone, near the outlet the velocity is negative shown by blue colour. It means that at the bottom part the flow pattern is reversed which is required for the separation.

Fig. 4 shows the velocity vectors at transverse and longitudinal midsection of the cyclone. A clear circular orientation of velocity vectors reinforce the fact that a good swirling motion of fluid is developed which, as will be seen later, would give better collection efficiency.

Fig. 5 shows the streamlines of randomly selected 15 particles leaving both the outlets. It clearly shows that out of the selected particles most of them are leaving from the bottom outlet with only a few leaving from the top. This indicates the higher collection efficiency of the cyclone for these randomly selected particles.

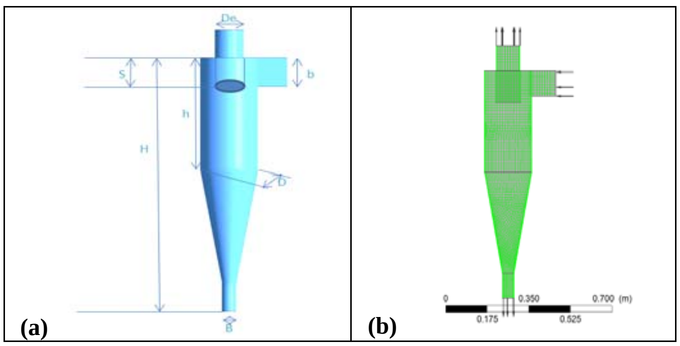

Fig. 1: Schematic presentation of (a) cyclone geometry and (b) grid.

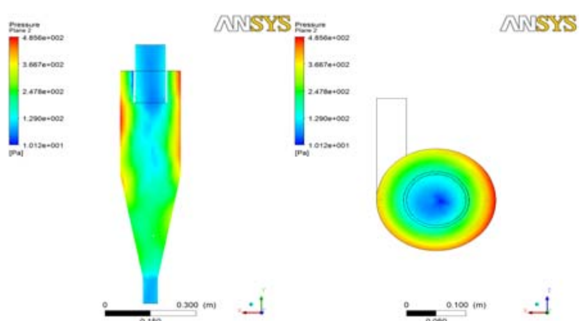

Fig. 2: Contour of pressure

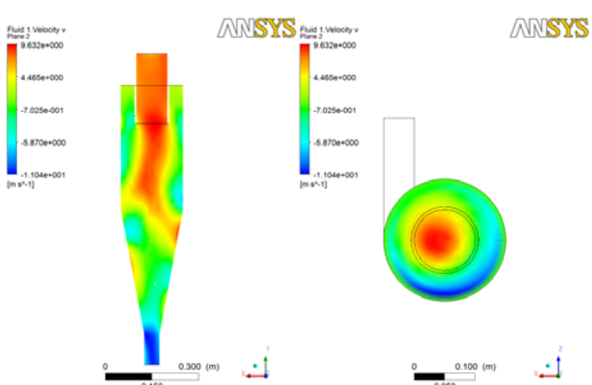

Fig. 3: Contour of axial velocity

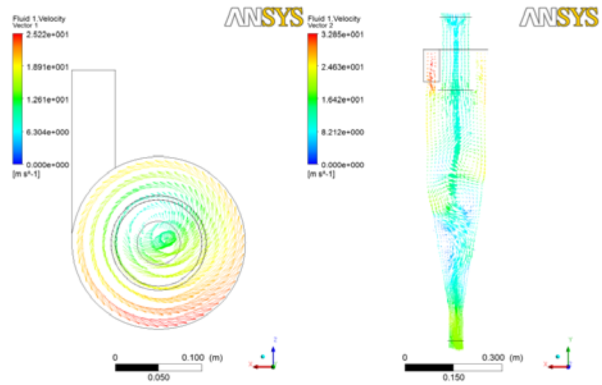

Fig. 4: Velocity vectors

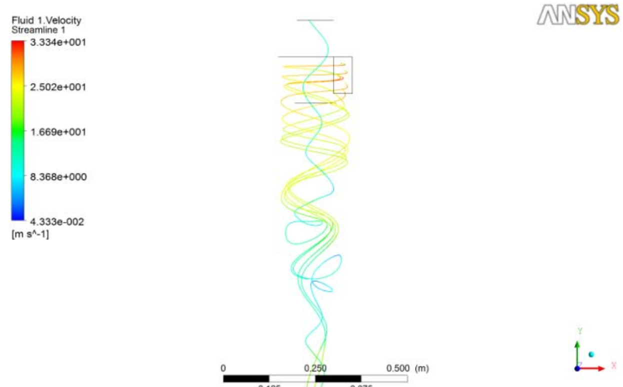

Fig. 5: Streamlines of particles.

REFERENCES

[1] Wang, B., D. L. Xu, G. X. Xiao, K. W. CHU And A. B. YU , "Numerical Study of Gas-solid Flow in a Cyclone Separator", Third International Conference on CFD in the Minerals and Process Industries ,CSIRO, Melbourne, Australia, (2003).

[2] Gimbun, Jolius, Thomas S. Y. Choong, A. Fakhru'l-Razi& T.G. Chuah , "Prediction of The Effect of Dimension, Particle Density, Temperature, and Inlet Velocity on Cyclone Collection Efficiency", JurnalTeknologi, 40(F), 37-50, (2004).

[3] Elsayed, Kh., and C. Lacor , "A CFD Study of the Effects of Cone Dimensions on the Flow Field of Cyclone Separators using LES", 13th International Conference on Aerospace Sciences & Aviation Technology, Asat- 13, 26 – 28, (2009).

[4] Marinuc, M. And F. Rus, "The Effect of Particle Size and Input Velocity on Cyclone Separation Process", Bulletin of the TransilvaniaUniversity of Braşov , series2 , Vol. 4 (53), 117-122, (2011).