

Evaluation of Preduster in Cement Industry Based on Computational Fluid Dynamic

E L Septiani^{1,a}, W Widiyastuti^{2,b}, A Djafaar³, I Ghozali³, H M Pribadi³

¹ Department of Chemical Engineering, Universitas Internasional Semen Indonesia (UISI), Kompleks PT. Semen Indonesia (Persero) Tbk., Gresik 61122 Indonesia

² Department of Chemical Engineering, Industrial Technology Faculty, Institut Teknologi Sepuluh Nopember, Surabaya 60111, Indonesia

³ Directorate of Production, PT. Semen Gresik (Persero) Tbk., Ds SumberArum, Kerek, Tuban 62356 Indonesia

^aeka.septiani@uisi.ac.id, ^bwidi@chem-eng.its.ac.id

Abstract. Ash-laden hot air from clinker in cement industry is being used to reduce water content in coal, however it may contain large amount of ash even though it was treated by a preduster. This study investigated preduster performance as a cyclone separator in the cement industry by Computational Fluid Dynamic method. In general, the best performance of cyclone is it have relatively high efficiency with the low pressure drop. The most accurate and simple turbulence model, Reynold Average Navier Stokes (RANS), standard k- ϵ , and combination with Lagrangian model as particles tracking model were used to solve the problem. The measurement in simulation result are flow pattern in the cyclone, pressure outlet and collection efficiency of preduster. The applied model well predicted by comparing with the most accurate empirical model and pressure outlet in experimental measurement.

1. Introduction

The highest heat loss as hot air in cement industries obtained from kiln have been used as heater media in calciner and multi-stage preheater with countercurrent flow. After heating raw materials from preheater, hot air still used to reduce water content of coal, fuel of calciner and kiln, in the coal mill. However, this hot air is existing ash content therefor separating process is needed first before sent to coal mill. Preduster, cyclone separator, has been used for removing large particles in many industries because it has low capital cost, very high duct loading, ability to operate at high temperature, and low maintenance requirement but it have low efficiency especially for small particles [1,2]. Therefore, evaluation of preduster efficiency is needed. Many researchers have been looking for cyclone efficiency both in operating cost and collection efficiency by experiment or theoretic. Chuah et. al. [3] compared experimental and theoretical pressure drop in cyclone by Shepherd and Lapple (1939), Casal and Martinez (1984), Dirgo (1985) and Coker (1993) model. Their study shows Coker model predicted cyclone pressure drop, which is related to operating cost and collection efficiency. Bogodage and Leung [4] estimated overall collection efficiency of cyclone by experimental method, then improve the cyclone separator performance by variation of down-comer tubes. Nevertheless, these method are not feasible to predict cyclone efficiency in industrial scale due to manufacturing scale-up parameter and operation cost. Now a adays, theoretical cyclone efficiency widely evaluated by Computational Fluid Dynamics (CFD) method after the first CFD investigation presented by Boyson et. al.. They reported pressure drop that is predicted by CFD has a great agreement with measured data.



CFD offered momentum, heat, and mass transfer data to simulate cyclone operation which is based on combination of mass rotational forces, drag forces, and other forces [5]. Brennan et. al [6] validated performance of industrial cyclone separators by CFD modelling. Their study compared Differential Reynold Stress (DRSM), Large Eddy Simulation (LES), and Laminar flow model. They reported that DRSM and LES model predicted well. Kepa [7] also reported that CFD modelling allows the analysis of large-size cyclones that are very difficult to analyze in experimental studies by RSM turbulence model. Since LES and RSM method burdened with a lot of computational effort [8-10] although let the result that closed to experimental measurement, Reynold Average Navier Stokes (RANS), the simpler model gave accurate estimation [11], may be considered to provide this present study. For particles tacking is defined by Lagrangian calculation that have been commonly used. Since collection efficiency is affected by velocity inlet [12] and the rate of hot air is can't be changed, here the influence of inlet dimension in Le/D ratio to pressure drop and collection efficiency of the preduster was examined.

2. Methodology

Preduster performance in this study was calculated by commercial CFD, ANSYS FLUENT 16.2. The geometry was defined in 3-D calculation domain. Figure 1(a) shows the sketch of preduster dimension in millimeter size.

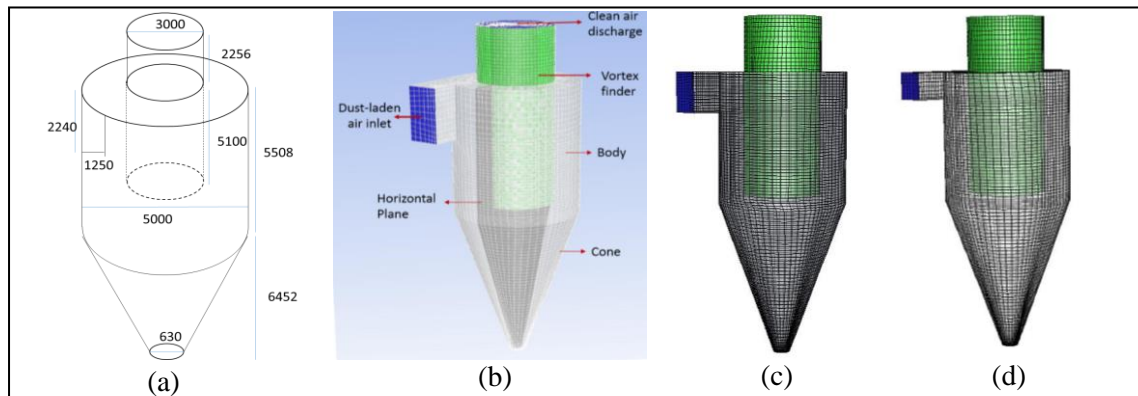


Figure 1. (a) Preduster geometry, (b) Name section and Preduster grid of Le/D = 0.5 (current preduster), (c) Peduster grid of Le/D = 0.3, (d) Preduster grid of Le/D = 0.2.

The geometry was defined by Workbench Design Modeller software then should be divided by Meshing software into several grids. Here, the grids of current preduster were generated in 84,148 grids with skewness 0.153 as can be seen at Fig. 1(b). While for Le/D=0.3 was divided by 85,734 meshes with skewness 0.159 and 84,914 grids with skewness 0.158 for Le/D=0.2. Its mean the calculated domain has relatively high quality. Reynold Average Navier Stokes model is used to describe turbulence model by assuming fully turbulence flow. This model solved two equations [5,13], turbulence kinetic energy (k) and dissipation rate (ϵ), here

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (2)$$

Where $G_k = -\rho \overline{u_i u_j} \frac{\partial u_j}{\partial x_i}$, eddy viscosity calculated by $\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \mu_t$. C_μ is constant 0.09. $C_{1\epsilon}, C_{2\epsilon}$ are

1.44; 1.92, $\sigma_k, \sigma_\epsilon$ are Prantdl Number 1, 1.3. S_k, S_ϵ are user defined source. Meanwhile, the particles movement was defined by Lagrangian model, Discrete Phase Model (DPM), which solved that force balance of particle is

$$\frac{du_p}{dt} = F_D (\bar{u} - \bar{u}_p) + \frac{\bar{g}(\rho_p - \rho)}{\rho_p} + \bar{F} \quad (3)$$

\vec{F} is additional acceleration, $F_d(\vec{u} - \vec{u}_p)$ is drag force each particle mass, and F_d presented by $F_d = \frac{18\mu C_d \text{Re}}{\rho_p d_p^2} \cdot 24$. While \vec{u} and \vec{u}_p are fluid velocity and particle velocity [5]. Contour of pressure in

the preduster was observed from horizontal plane that shown at Fig. 1(b). Overall collection efficiency was calculated by

$$\eta_o = \sum \eta_j m_j \quad (4)$$

$$\eta_j = \frac{1}{1 + \left(\frac{d_{pc}}{d_{pj}}\right)^2} \quad (5)$$

$$d_{pc} = \left[\frac{9\mu W}{2\pi N e V_i (\rho_p - \rho_s)} \right]^2 \quad (6)$$

m_j is mass fraction of particles in the j th particle size range. d_{pj} is characteristic diameter of the j th particle size range (in μm). N_e is number of effective turns and V_i is velocity inlet.

3. Result and Discussion

The result of this work indicates well prediction. It can be seen in Fig. 2(a), the air and particles through inlet of preduster entered from cyclone top. This dust-laden air flow in a downward spiral against the cyclone body's outer wall, that is called outer vortex while particles were separated by centrifugal force. The outer vortex gets smaller as the air flows downward along the cone. Furthermore, the vortex hopped up in smaller spiral flow, it is called inner vortex, leaves the preduster to outlet in the top by passing the vortex finder.

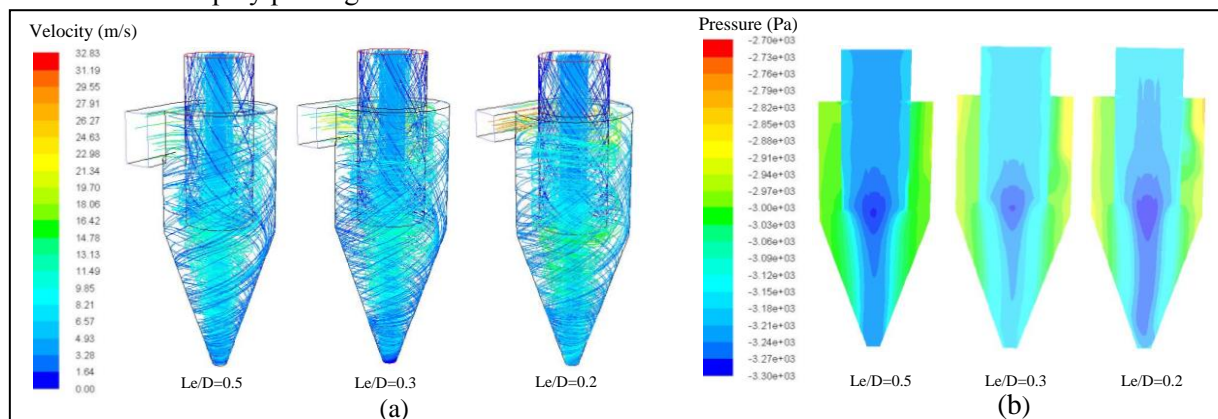


Figure 2. (a) flow pattern of dust-laden air in the preduster by velocity (m/s) path line, (b) Pressure contour in Horizontal plane (Pa)

Figure 2(b) shows similarly approximated pressure outlet to measured pressure in plant. CFD method predicted pressure outlet in current preduster -3205 Pa, while the actual pressure inlet of hot air inlet in the coal mill is -2546.5 Pa. this difference may caused by the number of geometry mesh, besides the friction losses by elbow and pipe which have length about over 10 meters. The pressure drop in CFD method give well comparison with the most accurate empirical model, Shepherd and Lapple method, as can be seen in Fig. 3(a). Increasing velocity inlet due to reducing Le/D ratio will be rise the pressure drop in the preduster.

Ash particles content in hot air were being approached by Rosin-Rammler distribution in particle range between 0.5 μm and 25 μm with 10% of hot air flow rate. particles passed both in clean air discharge and collected dust discharge was calculated in steady state condition. From this, the over-all collection efficiency calculation for current preduster using eq. (4) is 47.59%. This relatively low efficiency may be affected by the dimension of preduster so that the particles also swirl in the inner vortex from the bottom to the gas outlet and the largest number of particles loading in which smaller than 5 μm . Whereas decreasing Le/D ratio to 0.3 increase the overall efficiency until 66.95%, it can

be caused by some particles go down directly without flowing in the inner vortex from the bottom as seen as Fig. 3(b). however, reducing Le/D ratio to 0.2 did not increase overall collection efficiency because the particles being trapped in the swirling hot air flow in the preduster.

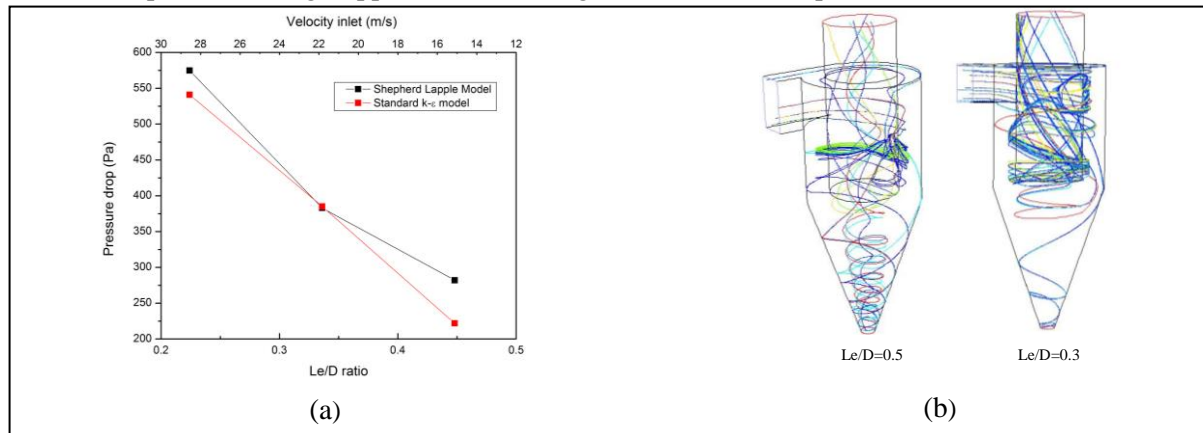


Figure 3. (a) Pressure drop comparison between Shepherd and Lapple and Standard $k-\epsilon$ method, (b) Particle tracking in the preduster

4. Summary

This present study proved that CFD can predicted the performance of cyclone well that is shown by flow pattern of air containing dust, pressure outlet approximation in current preduster and good comparison pressure drop to Shepherd and Lapple model. The overall collection efficiency was relatively low for the current preduster. Increasing velocity due to Le/D ratio of 0.3 rise the overall collection efficiency, however reducing Le/D to 0.2 drop the overall collection efficiency.

Acknowledgement

Research grant allowed by LPPM-UIIS is gratefully acknowledged. PT Semen Indonesia (Persero) Tbk. Is also acknowledged for providing necessary data.

References

- [1] Y. Kosaki, T. Hirai, Y. Yamanaka, K. Takeshima. 2015 *J. Powder Technology* 277 p. 22–35
- [2] F. Ficici, V. Ari, M. Kapsiz. 2010 *International Journal of the Physical Sciences* Vol. 5 (6), p. 804–813
- [3] T. G. Chuah, J. Gim bun, T. S. Y. Choong. A. Fakhrulrazi. 2003 *J. Chem. Eng. and Environmental* Vol. 2, No.2, p 67 – 71
- [4] S. G. Bogodage and A. Y. T. Liung 2016 *J. Hazardous Material* 311 p. 100 – 114.
- [5] Anonim, 2011 *ANSYS FLUENT 14.0 Teory Guide* (Canonsburg: ANSYS Inc.)
- [6] M. Brennan, P. Holtam, M. Narasimha. 2014 *International journal of Minerals Process* 133 p.1–12
- [7] Kepa. 2013 *J. Separation and Purification Technology* 118 p. 105–111
- [8] G. Wan, G. Sun, X. Xue, M. Shi. 2008 *J. Powder Technol.* 183, 94.
- [9] J.J. Derksen, H.E.A. van den Akker, S. Sundaresan. 2008 *AIChE J.* 54 (4) 872.
- [10] F.J. Souza, R.V. Salvo, D.A.M. Martins. 2012 *J. Separat. Purificat. Technol.* 94, 61.
- [11] E. L. Septiani, W. Widiyastuti, S. Winardi, S. Machmudah, T. Nurtono, and Kusdianto. Effect of turbulence modelling to predict combustion and nanoparticle production in the flame assisted spray dryer based on computational fluid dynamics. 2016 *Padjadjaran International Physics Symposium 2016. AIP Conference Proceedings*, 1712, 040002
- [12] Sgrott Jr., O. L., Noriler, D., Wiggers, V. R., Meier, H. F. 2015 *J. Powder technology* 2015.02.039
- [13] Versteeg, H. K. and Malalasekera W. 2007 *An introduction to computational fluid dynamics* (England: Pearson Education Limited)