Australian Journal of Basic and Applied Sciences, 8(4) Special 2014, Pages: 521-527

AENSI Journals

Australian Journal of Basic and Applied Sciences

ISSN:1991-8178

Journal home page: www.ajbasweb.com

CFD Simulation of Dust Cloud Formation in Silo

[1,2]S.I.Rani, [2,3]J.Gimbun, [2]B.A. Aziz

[1]*Faculty of Chemical Engineering Technology, TATI University College, 24000Terengganu, MALAYSIA.*
[2]*Faculty of Chemical Engineering & Natural Resources Engineering,* University of Malaysia Pahang, *26300 Kuantan, MALAYSIA.*
[3]*Centre of Excellent for Advanced Research in Fluid Flow (CARIFF),* University of Malaysia Pahang, *26300 Kuantan, MALAYSIA.*

ARTICLE INFO

Article history:
Received 20 November 2013
Received in revised form 24 January 2014
Accepted 29 January 2014
Available online 5 April 2014

Key words:
Dust cloud; flow field; RNG; DPM; silo

ABSTRACT

Silo feeding process via pneumatic transport pipe will generates the initial turbulence that causes the dispersion of particles in air. This phenomenon is known as dust cloud formation which is likelihood for an explosive atmosphere. Thus, understanding of dust cloud formation in silo is important for safety measures. This paper presents the results of flow field, in terms of mean and root mean square (RMS) velocity of dust cloud in a silo during axial feeding using commercial computational fluid dynamics (CFD) code, Fluent. The influence of grid size, modeling approach and discretization schemes were examined using turbulence model namely renormalization group k-ε (RNG) combined with disperse phase model (DPM) which follows Eulerian-Lagrangian (E-L) approach. The E-L approach treats solid particles as moving points in the computational domain and their movement is tracked as they moved through the gas flow. Two-way coupling effect was considered since the continuous phase flow field is impacted by the discrete phase. The profiles of mean and RMS velocity at five different axial locations Z = 750 mm, 1750 mm, 2750 mm, 3750 mm and 4750 mm were examined to recognized the flow pattern inside silo. Predicted mean and RMS velocity using combination of DPM and RNG turbulence model with various discretization and pressure interpolation schemes were plotted and compared against experimental measurement adopted from literature, showing a plausibly good agreement with second order upwind scheme and standard pressure interpolation scheme. It was found that predicted flow field has a great influence to the silo height. Higher mean velocity in downward direction was predicted in the center region closer to the top of the silo due to gravitational force on the particles. Moreover, it was found that turbulence flow (RMS velocity) is correspondingly increase with increasing the axial positions, indicating that region closer to the feeding inlet favor the initial turbulence formation. This suggests that combination of RNG turbulence model and DPM could be employed in dust cloud formation study.

To Cite This Article: S.I.Rani, J.Gimbun, B.A. Aziz, CFD Simulation of Dust Cloud Formation in Silo. ***Aust. J. Basic & Appl. Sci.,*** *8(4): 521-527, 2014*

INTRODUCTION

Silo is an industrial enclosure that is used to store bulk materials with the volume ranges from a few cubic meters to some thousand cubic meters. Commonly, silos are used for bulk storage of agricultural dusts, agricultural products, carbonaceous dusts, chemical dusts and plastic dusts. However in reality, about 80% of bulk materials are combustible (Krause, 2009). Hence, fire and explosion is the potential hazard for industrial plants with dust processing equipment, particularly to silo. Owing to their large volume, the consequences of fire and explosion in silos are usually devastating. When the silos are interconnected either directly or via the solids handling equipment, the explosion can propagate to the other silo and the consequences would be more severe. For instance, in 2008, a sugar dust fire and explosion occurred in Port Wentworth, Georgia, which caused 14 fatalities and 36 injuries and demolished the silos and other process equipments (CSB, 2009).

During pneumatic conveying with axial release into silos, the fine particles are dispersed in air may produce explosive dust clouds that pose a major risk to the industries involves in manufacturing, using and/or handling bulk powders. Typically, the initial turbulence is generated by the feeding process from an equipment prior to the silo whether a cyclone, a pneumatic transport pipe, a mixer, a bucket elevator or a mill. Eventually, explosive dust cloud may be formed inside the silo by this initial turbulence, which may trigger an explosion-induced turbulence.

Although many phenomenon of gas-solid flow are still not well known during silo feeding, Hauert and Vogl (1995) suggested that the measurements of flow field and dust distribution are able to illustrate the real dust clouds in industrial enclosures. However, the present work will consider only the initial turbulence since it is a trigger feature for dust explosion. Experimental investigation of dust cloud formation in a silo requires costly instrument

Corresponding Author: S.I. Rani, Department of Chemical Engineering Technology, Faculty of Chemical Engineering Technology, TATI University College, 24000 Terengganu, MALAYSIA.
E-mail: ilyani@tatiuc.edu.my

such as gamma-ray tomography and particle image velocimetry besides being time consuming. Consequently in this present study, emphasize was put on Computational Fluid Dynamics (CFD) simulation as it could portrays an insight view of the process. Our previous study found that the Renormalization Group k-ε (RNG) has shown the best prediction for gas-solid flow than standard k-ε (SKE) and Realizable k-ε (RKE) (Rani *et al*., 2013). Therefore, the objectives of this study are to evaluate the performance of RNG turbulence model for predicting the initial turbulence in a silo and to investigate the profiles of mean and RMS velocity at different axial levels inside the silo in order to better understand the flow patterns.

2. Computational Approach:

2.1. Description of the case problem:

Fig. 1 depicts a cylindrical silo considered in this work. The silo has a height of 5 m and a diameter of 1.6 m, which was used in experimental studied by Hauert and Vogl (1995). Three exhaust air filters were mounted at the silo top. Cornstarch with the mean diameter of 15µm and the pneumatic axial feeding rate of 3 kg/m^3 from the top of the silo has been simulated following the experimental work by Hauert and Vogl (1995). The feeding velocity is 23 m/s through the 75 mm inner diameter of conveying pipe located at the center of silo.

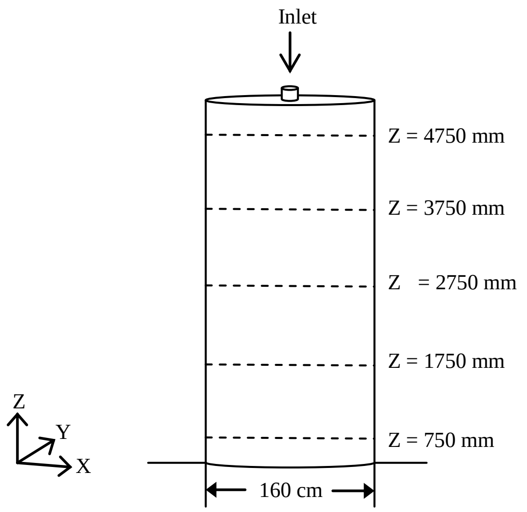

Fig. 1: Cylindrical silo geometry with various axial positions.

2.2. Modeling gas phase:

The gas phase was modelled using the classical turbulence model which provides a major role in precisely predicting the gas-solid flow. For instant, we consider the application of RNG turbulence model which are based on Reynolds Average Navier-Stokes (RANS) equations (time-averaged). The turbulence kinetic energy, *k* and the dissipation rate, *ε* are obtained from the following equations:

$$\underbrace{\frac{\partial(\rho k)}{\partial t}}_{\text{time derivative}} + \underbrace{\frac{\partial}{\partial x_i}(\rho u_i k)}_{\text{convection}} = \underbrace{\frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right)}_{\text{diffusion}} + \underbrace{\rho P_k}_{\text{production}} - \underbrace{\rho\varepsilon}_{\text{destruction}} \quad (1)$$

and

$$\underbrace{\frac{\partial(\rho \varepsilon)}{\partial t}}_{\text{time derivative}} + \underbrace{\frac{\partial}{\partial x_i}(\rho u_i \varepsilon)}_{\text{convection}} = \underbrace{\frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right)}_{\text{diffusion}} + \underbrace{S_\varepsilon}_{\text{source term}} \quad (2)$$

The turbulent (eddy) viscosity, μ_t, is obtained from:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (3)$$

The relation for production term P_k, for the *k-ε* variant model is given as:

$$P_k = \mu_t \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \frac{\partial u_j}{\partial x_i} \tag{4}$$

The RNG have significant changes in the ε equation and improves the ability to model highly strained flows, predicting swirling flow and Reynolds number flows. RNG has an additional term in the ε transport equation, besides providing an analytical formula for the turbulent Prandtl numbers derived using RNG theory. Thus the source term S_ε for RNG is given by:

$$S_\varepsilon = \rho \left(C_{1,RNG} \frac{\varepsilon}{k} P_k - \sigma^{-1} \frac{\varepsilon^2}{k} - C_{2,RNG} \frac{\varepsilon^2}{k} \right) \tag{5}$$

Where σ^{-1} is the inverse effective Prandtl number given by:

$$\sigma^{-1} = \frac{C_\mu \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3} \tag{6}$$

Instead of constant value for turbulent Prandtl number in k-ε, it is provided analytically in RNG by the following equation:

$$\left| \frac{\sigma^{-1} - 1.3929}{\sigma_0^{-1} - 1.3929} \right|^{0.6321} \left| \frac{\sigma^{-1} - 2.3929}{\sigma_0^{-1} - 2.3929} \right|^{0.3679} = \frac{\mu_{mol}}{\mu_{eff}} \tag{7}$$

Where σ_0^{-1} = 1.0. In the high Reynolds number limit ($\mu_{mol}/\mu_{eff} << 1$), the inverse turbulent Prandtl number is $\sigma_k^{-1} = \sigma_\varepsilon^{-1} \approx 1.393$. Similar to the RKE model, $\eta = Sk/\varepsilon$, and $S = \sqrt{2 S_{ij} S_{ij}}$ is a modulus of mean rate of strain tensor. The model constants are η_0= 4.38, β = 0.012, $C_{\varepsilon 1}$ = 1.42, $C_{\varepsilon 2}$ = 1.68, and $\sigma_k = \sigma_\varepsilon$ = 1.39 (Yakhot and Orszag, 1986).

2.3. Modeling Discrete Phase:

The discrete phase model follows Eulerian-Lagrangian (E-L) model; which treated the particles as non-continuous phases. This is done by integrating the force balance on the particles and is given as:

$$\underbrace{\frac{\partial u_P}{\partial t}}_{\text{Acceleration}} = \underbrace{F_D (u - u_P)}_{drag\ \ force} + \underbrace{\frac{g_x (\rho_P - \rho)}{\rho_P}}_{gravity\ \ force} + \underbrace{F_x}_{other\,\text{forces}} \tag{8}$$

$$F_D = \frac{18\mu}{\rho_p D_p^2} \frac{C_D \operatorname{Re}}{24} \tag{9}$$

$$\operatorname{Re} = \frac{\rho D_p |u_p - u|}{\mu} \tag{10}$$

$$C_D = a_1 + \frac{a_2}{\operatorname{Re}} + \frac{a_3}{\operatorname{Re}^2} \tag{11}$$

The constant are given by Morsi and Alexander (1972) since the injected particles are spherical. The Saffman's lift forces are also considered as follows:

$$\vec{F} = \frac{2 K v^{1/2} \rho d_{ij}}{\rho_p d_p (d_{lk} d_{kl})^{1/4}} (\vec{\upsilon} - \vec{\upsilon}_p) \tag{12}$$

where, the constant K = 2.594 and d_{ij} is a deformation tensor.

3. Modeling Approach:

The dust-air flow inside the silo were discretized and solved iteratively by using commercial CFD code, Fluent 6.3.26. Prior to the simulation, Gambit 2.4.6 was used to draw the 3D configuration consists of 100% high quality hexahedral grid. The semi-implicit method for pressure linked equation (SIMPLE) method was used for the pressure-velocity coupling and various discretization upwind schemes such as the first order and second order, as well as the standard and pressure staggering option (PRESTO) interpolation schemes. The RNG turbulence model was employed in the simulation. The discrete phase model (DPM) which follows the Eulerian-Lagrangian approach was used to model the particle flow. In most industrial flows, the particles are highly inertial and their effect on the fluid turbulence cannot be ignored. Therefore, the gravitational force and Saffman’s lift force were applied. In the present study, air was the continuous phase and solid particles were the dispersed phase. Hence, to ensure the mutual contact between continuous and disperse phase, two-way coupling effect were employed. The inert particles were used for initial conditions and the particles were injected from the surface according to Rosin-Rammler logarithm diameter distribution. Only unsteady solvers were executed in this work. Convergence was assumed when the residual of the equations decreased more than three orders of magnitude.

4. Grid Dependent Study:

The grid dependent study was performed for three different grid sizes, i.e. coarse (121000), medium (363000) and fine (521000) to evaluate the suitability of the prepared grid. As presented in Fig. 2, negligible differences were observed from the fine and medium grid, and hence the medium grid (363000) was preferred for the remainder of this work to minimize the computational time.

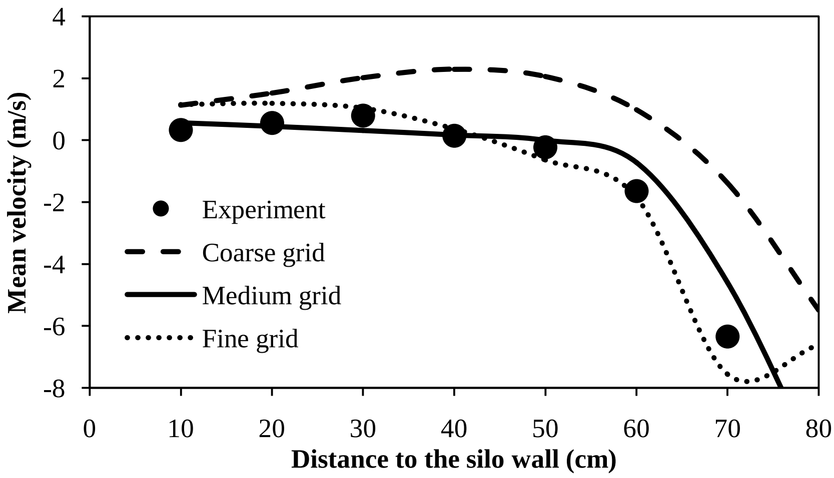

Fig. 2: Results of grid dependent study.

RESULT AND DISCUSSION

Results from the CFD simulation were compared to the experimental data measured by Hauert and Vogl (1995) at 3750 mm from the bottom. This study emphasis only on unsteady solver since the experimental measurement represents average value of instantaneous quantity which can only be represented correctly by the unsteady simulation (Rani *et al*., 2013). The measured and predicted mean velocity using various discretization techniques and pressure interpolation schemes are presented in Fig. 3(a). As can be seen, the predicted mean velocity using second order upwind scheme and standard pressure interpolation decreases to the center of the silo, which shows similar trend as experiment. This is attributed to the reduction of numerical diffusion for higher order discretization such as second order upwind scheme. Besides, the mean velocity was found to be influenced by feeding process since 8000 particles were fed continuously through the pneumatic transport pipe. Thus, high flow velocity region was predicted in negative value as a result of gravitational force dominated. Both predicted and measured mean velocities were about 0 m/s at 10 cm to the silo wall. This is due to downward movement of particles and upward movement of air to exhaust as the silo filled. PRESTO scheme portrays a poor prediction in this work because they are mainly favorable for a strong swirling flow such as cyclone, while the flow in a cylindrical flat bottom silo does not have such features, although the air movement to the exhaust filters mounted at the silo top may induce more chaotic particle movement.

The calculated mean velocity at different axial levels inside the silo is presented in Fig. 3(b). The predicted mean velocities are decrease towards the center of the silo. Closer to the wall, the mean velocities are pointing in upward direction (positive values); however, the contrary flows can be observed in the center region, where the velocities are in downward direction (negative values), due to gravitational force on the particles. At the position closer to the particles feed inlet (e.g. z ≥ 3750 mm), the velocity at 10 cm from the silo wall ranges from 0 to 0.5 m/s upward and the highest mean velocity (≈ 2 m/s) is found closer to the bottom of the silo (z = 750 mm). This is due to gas re-circulating flow to the exhaust air filter at the top of the silo.

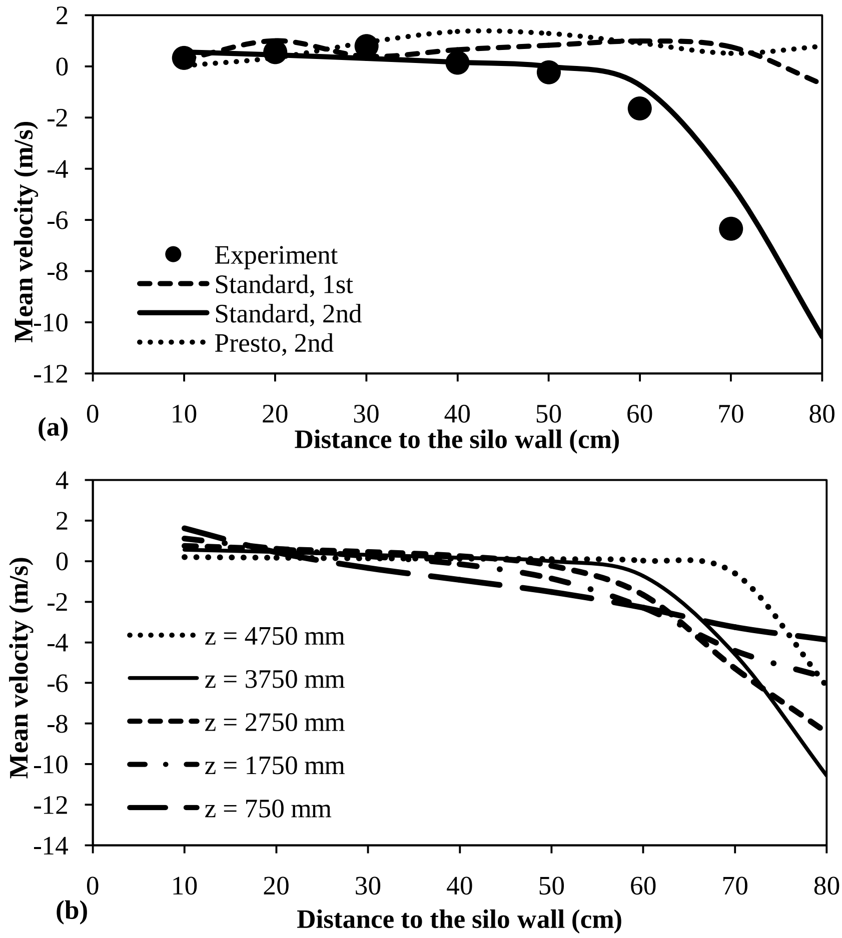

Fig. 3: Prediction of mean velocity for (a) various discretization techniques and (b) various axial levels inside the silo.

The predicted RMS velocity using RNG turbulence model and at various axial levels inside the silo are presented in Fig. 4. The RMS velocity was simulated using second order upwind scheme and standard pressure interpolation only, due to an expensive computational effort and time constraints. As can be seen in Fig. 4(a), the reverse trend found in predicted RMS velocity, which is in qualitative agreement with the experiment. The maximum velocity is found in the center of the silo in positive value since in present context, turbulence flow may be described by the movement of particles in dust clouds relative to each other in three dimensions. At the silo wall, the particles will lift up and the velocity reverses. Therefore, the turbulence took its maximum value at the centre of the silo. Thus, the RNG turbulence model is favourable to characterize dust cloud formation in silo as it is improved to gratify the strain flows and swirling flows analysis.

Predicted RMS velocity at various axial levels also portrays a similar trend as shown in Fig. 4(b). Closer to the feed inlet (e.g. z = 4750 mm), the particle movement induces intense turbulence with the highest RMS velocity (more than 18 m/s) is predicted at the center region of the silo. The lowest velocity fluctuation (≤ 2 m/s) is found at the bottom of the silo (z = 750 mm) and low RMS velocities at all axial positions (in the range of 0.5 to 1 m/s) are found at 10 cm from the silo wall due to particles built up as dust deposits or layer at the bottom of the silo. This suggests that, in this case, the gas re-circulating flow did not have great influence on the turbulence flow away from the feed position, e.g. closer to the silo wall and at the bottom of the silo. Thus, turbulence induced by dust feeding process is more likely to be produced closer to the feed inlet.

Australian Journal of Basic and Applied Sciences, 8(4) Special 2014, Pages: 521-527

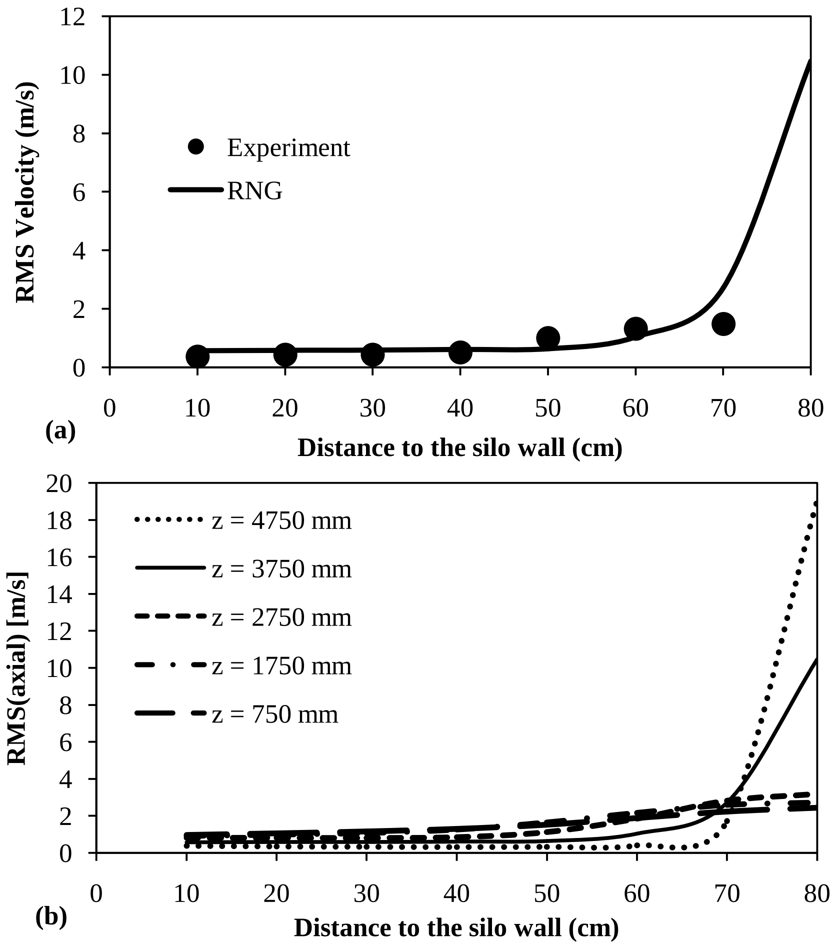

Fig. 4: Prediction of RMS velocity for (a) second order upwind scheme and standard pressure interpolation scheme and (b) various axial levels inside the silo.

Conclusion:

In this paper, a computational fluid dynamic (CFD) simulation of dust cloud formation during axial feeding process in a silo was attempted. The main purpose was to validate predictions of flow filed (mean velocity and RMS velocity) using RNG turbulence model with various discretization techniques and pressure interpolation schemes in combination with disperse phase model which follows Eulerian-Lagrangian approach. Besides, the effect of axial silo positions on the dust cloud formation during the feeding process was also investigated. It was found that the types of discretization and pressure interpolation schemes have major effect on mean velocity in the silo. The predicted values of mean velocity using second order upwind scheme and standard pressure interpolation scheme were found to be in agreement with the experimental data, due to intrinsic nature of feeding process. The overall trend of the modeling approach prediction agrees well with the experimental data adopted from literature. It was also revealed that the dust cloud formation is strongly depends on the silo height: the mean velocity at the center of the silo decreased as the height of the silo increased and the highest turbulence flow (RMS velocity) was predicted at the region closer to the feeding process. Results from this work may be useful to characterize dust concentration distribution and determine an ignition sensitivity of dust cloud in a silo, and hence may be employed as risk analysis tools

REFERENCES

CSB, 2009. “Investigation Report. Sugar Dust Explosion and Fire, Imperial Sugar Company”, US Chemical Safety and Hazard Investigation Board.

Hauert, F. and A. Vogl, 1995. “Measurement of Dust Cloud Characteristics in Industrial Plants, Final Technical Report: Protecting People, Equipment, Buildings and Environment Against Dust Explosion-CREDIT Project”, PL 910695.

Krause, U., 2009. “Introduction”, In Fires in Silos Hazards, Preventions and Fire Fighting, ed. Krause, U. Weinheim, Germany: Wiley-VCH Verlag GmbH & Co. KGaA. pp: 1-11.

Morsi, A.J. and S.A. Alexander, 1972. “An investigation of particle trajectories in two-phase flow system”, J. Fluid Mech., 55: 193-208.

Rani, S.I., J. Gimbun and B.A. Aziz, 2013. “Modeling of Gas-Solid Turbulence Flow in Silo”, Proceedings of the 6th International Conference on Process Systems Engineering (PSE ASIA 2013), pp: 784-789.

Yakhot, V. and S.A. Orszag, 1986. “Renormalization Group Analysis of Turbulence . I . Basic Theory”, Journal of Scientific Computing, 1: 13-51.