Proceedings of the 6th International Conference on Process Systems Engineering (PSE ASIA)
25 - 27 June 2013, Kuala Lumpur.

Modeling of Gas-Solid Turbulence Flow in Silo

S.I Rani,[a,b],J. Gimbun,[b,c], B.A. Aziz[b]

[a]*Faculty of Chemical Engineering Technology, TATI University College, Teluk Kalong, 24000 Kemaman, Terengganu, Malaysia.*
[b]*Faculty of Chemical Engineering & Natural Resources Engineering, University Malaysia Pahang, Tun Razak Highway, 26300 Gambang, Pahang, Malaysia.*
[c]*Center of Excellent for Advanced Research in Fluid Flow (CARIFF), University Malaysia Pahang, Tun Razak Highway, 26300 Gambang, Pahang, Malaysia.*

Abstract

This paper presents a computational fluid dynamics (CFD) study of gas-solid turbulence flow in a silo. The simulations were performed using three different turbulent models namely, standard k-ε (SKE), RNG k-ε (RNG) and Realizable k-ε (RKE). The gravitational force, Saffman lift force and two-way coupling effect were employed. The influence of grid size, discretization, modeling strategies and solver were examined. An Eulerian-Lagrangian approach was used to model the particle flow using Disperse Phase Model (DPM). The predicted gas-solid flow patterns inside the silo chamber were found to be in good agreement to the experimental data adopted from literature. Good predictions were achieved using RNG k-ε with discrepancy around 4%.

Keywords: CFD; Flow; Silo; Turbulence model; Disperse phase model.

1. Introduction

The dispersion of dust particles in air may produce explosive dust clouds that pose a significant risk to the industries involves in manufacturing, using and/or handling bulk powders. In fact, about 80% of bulk materials are combustible (Krause, 2009). This implies that the majority of industrial plants with dust processing equipment are susceptible to dust explosions. Industrial silos are used for storing bulk materials (dusts) with the volume ranges from a few cubic meters to some thousand cubic meters. Owing to their size, the consequences of fire and explosion in silos can be devastating. It would be more severe when the silos are interconnected; either directly or via the solids handling equipment where the explosion can propagate from one silo to the other. For instance, in 2008, an explosion triggered by a welding job in the silo pit occurred at grain and flour milling factory in Perak, Malaysia, which caused four fatalities, two injuries and severe damage to jetty structure, conveyor and other facilities (Ke, 2009).

During silo filling, dust cloud may be formed by turbulence flow which may trigger a turbulence-induced explosion. The turbulence flow responsible for the dust cloud is often induced by other equipment prior to the silo such as cyclone, pneumatic transport pipe, mixer, bucket elevator or bag filter. Experimental investigation of dust cloud characterization in a silo requires costly instrument such as particle image velocimetry and gamma-ray tomography besides being time consuming. Alternatively, Computational Fluid Dynamics (CFD) can provide a good prediction of gas-solid turbulence flow inside the silo chamber. Thus, this work aims to formulate a suitable modelling strategy for gas-solid flow in a silo and to evaluate the performance of various turbulence models namely standard k-ε (SKE), RNG k-ε (RNG) and Realizable k-ε (RKE) for predicting the flow pattern in a silo.

2. Computational Approach

2.1. Description of the case problem

The gas-solid flow in 12 m^3 cylindrical silo with a diameter of 1.6 m and height of 5 m as illustrated in Figure 1, which was experimentally studied by Hauert and Vogl (1995) were considered in this work. 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.

2.2. Modeling turbulence

Turbulence models play a significant role in accurately predicting both the gas and particle flows.   For instant, we consider three applications of classical model which are based on Reynolds Average Navier-Stokes (RANS) equations (time-averaged). For standard *k-ε* model, the *k* and *ε* equations are:

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

and

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

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

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{3}$$

The relation for production term P_k, for the *k-ε* variant models (i.e. SKE, RKE and RNG) is given as:

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

For the SKE model the source term, S_ε,is given by:

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

The model constants are: $C_{\varepsilon 1}$ = 1.44, $C_{\varepsilon 2}$ = 1.92, C_μ = 0.09, σ_k = 1.0 and σ_ε = 1.3 (Launder and Spalding, 1974).

This model is widely used despite the known limitation of the model. The SKE model performs poorly for complex flows involving severe pressure gradient, separation and strong streamline curvature. Improvements have been made to the model to improve its predictive capability leading to an introduction of its variants in RKE model. The RKE model allows certain mathematical constraint to be obeyed which ultimately improves the performance of this model. RKE different from SKE model in two ways; first it has a new formulation of turbulent viscosity and second it is employs a new transport equation for this dissipation rate. RKE model still has similar equation from SKE model, except the C_μ is no longer constant. The C_μ and S_ε for RKE are now given as:

$$C_\mu = \frac{1}{A_o + A_s \frac{U^* k}{\varepsilon}} \qquad (6)$$

where $A_s = \sqrt{6}\cos\phi$; $\phi = \frac{\cos^{-1}\left(\sqrt{6}w\right)}{3}, w = \frac{S_{ij}S_{jk}S_{ki}}{\tilde{S}^3}, \tilde{S}\sqrt{S_{ij}S_{ij}}$ and $U^* = \sqrt{S_{ij}S_{ij} + \tilde{\Omega}_{ij}\tilde{\Omega}_{ij}}$.

$$S_\varepsilon = \rho\left(C_1 S\varepsilon - C_2 \frac{\varepsilon^2}{k + \sqrt{\upsilon\varepsilon}}\right) \qquad (7)$$

The model constants are: A_0 = 4.0, C_2 = 1.9, σ_k = 1, and σ_ε = 1.2, where $C_1 = \max\left[0.43; \eta/\eta + 5\right]$; $\eta = Sk/\varepsilon$ and $S = \sqrt{2S_{ij}S_{ij}}$ (Shih et al., 1995).

RKE model offers largely the same benefits and has similar application as RNG. The RKE is a model possibly more accurate and easier to converge than RNG. However RNG have significant changes in the ε equation improves the ability to model highly strained flows. RNG model additional options aid in predicting swirling and flow Reynolds number flows. RNG differs from SKE because it 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:

$$S_\varepsilon = \rho\left(C_{1,RNG}\frac{\varepsilon}{k}P_k - \sigma^{-1}\frac{\varepsilon^2}{k} - C_{2,RNG}\frac{\varepsilon^2}{k}\right) \qquad (8)$$

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

$$\sigma^{-1} = \frac{C_\mu \eta^3 (1 - \eta/\eta_0)}{1 + \beta\eta^3} \qquad (9)$$

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

$$\left|\frac{\sigma^{-1} - 1.3929}{\sigma_0^{-1} - 1.3929}\right|^{0.6321}\left|\frac{\sigma^{-1} - 2.3929}{\sigma_0^{-1} - 2.3929}\right|^{0.3679} = \frac{\mu_{mol}}{\mu_{eff}} \qquad (10)$$

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 $S = \sqrt{2S_{ij}S_{ij}}$ 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 $\sigma_k = \sigma_\varepsilon$ = 1.393 (Yakhot and Orzag, 1986).

2.3. Modeling discrete phase

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

$$\underbrace{\frac{\partial u_P}{\partial t}}_{\text{Acceleration}} = \underbrace{F_D\left(u - u_P\right)}_{drag\ force} + \underbrace{\frac{g_x\left(\rho_P - \rho\right)}{\rho_P}}_{gravity\ force} + \underbrace{F_x}_{other\,forces} \qquad (11)$$

$$F_D = \frac{18\mu}{\rho_p D_p^2}\frac{C_D \,\mathrm{Re}}{24} \qquad (12)$$

$$\mathrm{Re} = \frac{\rho D_p\left|u_p - u\right|}{\mu} \qquad (13)$$

$$C_D = a_1 + \frac{a_2}{\mathrm{Re}} + \frac{a_3}{\mathrm{Re}^2} \tag{14}$$

Where 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:

$$\vec{F} = \frac{2Kv^{1/2}\rho d_{ij}}{\rho_p d_p (d_{lk} d_{kl})^{1/4}} \left(\vec{\upsilon} - \vec{\upsilon}_p\right) \tag{15}$$

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

2.4. Modeling strategy

In the present study, air was the continuous phase and solid particles were the dispersed phase. The gas-solid flow inside the silo were discretized and solved iteratively by using commercial CFD code, Fluent 6.3.26. Gambit 2.4.6 was used as pre-processing tool to build the 3D configuration consists of 100% high quality hexahedral grid. The 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 PRESTO interpolation schemes. Three different turbulence models namely the SKE, RNG and RKE, were 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 in this study, the gravitational force, the Saffman's lift force and two-way coupling effect were employed. The inert particle was used for initial conditions and the particle stream was injected from the surface according to Rosin-Rammler logarithm diameter distribution. Two different solvers namely steady and unsteady solvers were evaluated in this work. Results for the steady solver were obtained after the residuals fall below 10^{-4}. Meanwhile for unsteady solver data were taken as a statistical average from up to 1000 time steps after the pseudo convergence was achieved.

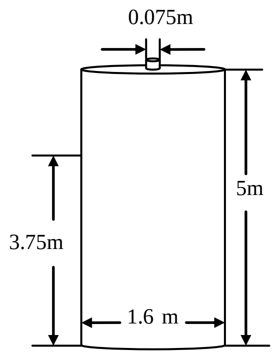

Figure 1. Cylindrical silo geometry

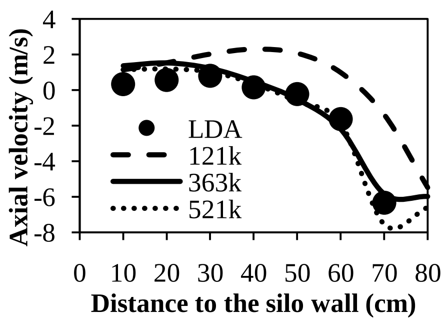

Figure 2. Grid dependent study

2.5. Grid dependent study

The grid dependent study was performed for three different grid sizes, i.e. coarse (121000), intermediate (363000) and fine (521000) to evaluate the suitability of the prepared grid. As shown in Figure 2, no significant differences were observed from the fine and intermediate grid, and hence the intermediate grid (363000) was chosen for the remainder of this work to minimize the computational effort.

3. Results and Discussion

Prediction from the CFD simulation was compared to the Laser Doppler Anemometer (LDA) measurement by Hauert and Vogl (1995) at 3.75 m from the bottom. The unsteady solver gives a closer agreement with experimental data as shown in Figure 3(a). Most of the published work in relation to CFD study of gas-solid flow only concern about the steady simulation and compared to the experimental data which is not exactly a steady-state data but instead a pseudo-steady from statistical point of view. Such an assumption may be acceptable when comparing a mean flow which is the statistical average of the flow, but it is not an ultimate solution. Since the experimental measurement represent average value of instantaneous quantity which can only be represented correctly by the unsteady simulation. Figure 3(b) shows the prediction of axial velocity obtained using difference discretization techniques. It was found that a combination of the second order upwind scheme and standard pressure interpolation give a better agreement to experimental data. This is attributed to the reduction of numerical diffusion for higher order discretization such as second order upwind scheme. PRESTO scheme did not give a good prediction in this work because they are mainly favorable for a strong swirling flow such as cyclone, while the flow in a cylindrical silo does not have such features, although the particle movement may induce a weak gas re-circulation.

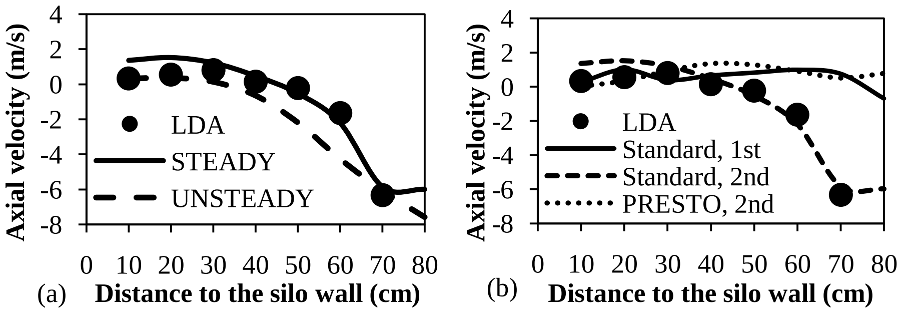

Figure 3. Prediction of axial velocity for (a) solver comparison, (b) various discretization techniques using RNG turbulence model.

In this present study it was found that the RNG model with standard pressure interpolation and second order upwind discretization scheme yields best results for both axial and RMS velocity as seen in Figure 4(c) and (d). Better prediction by RNG may be attributed by the inclusion of an improved statistically correct representation of turbulence within the model (Yakhot and Orzag, 1986). A weak re-circulating flow may present in a silo due to particle movement downward and upward movement of air to exhaust as the silo filled. Thus, the turbulence model that takes re-circulation into account as the RNG is more favorable. The SKE and RKE predictions on the mean and RMS velocity are acceptable; however, prediction of RMS velocity towards the center of the silo is not satisfactory.

4. Conclusion

CFD simulation of gas-solid flow in a silo has been successfully performed using both the Eulerian-Lagrangian and RANS turbulence models. All the RANS turbulence models employed in this work shows satisfactory agreement with the experimental data adopted from literature. However, the best agreement on mean and turbulence flow was achieved using the RNG turbulence model. It was also found that unsteady solver gave a better prediction on mean and turbulence flows due to intrinsic nature of turbulence which can only be predicted well by analyzing the data from unsteady solution. Findings from this work may be useful for development of a dust explosion model of a silo in the future which can be realized by enabling the particle-gas reaction, and hence may be employed as risk assessment tools.

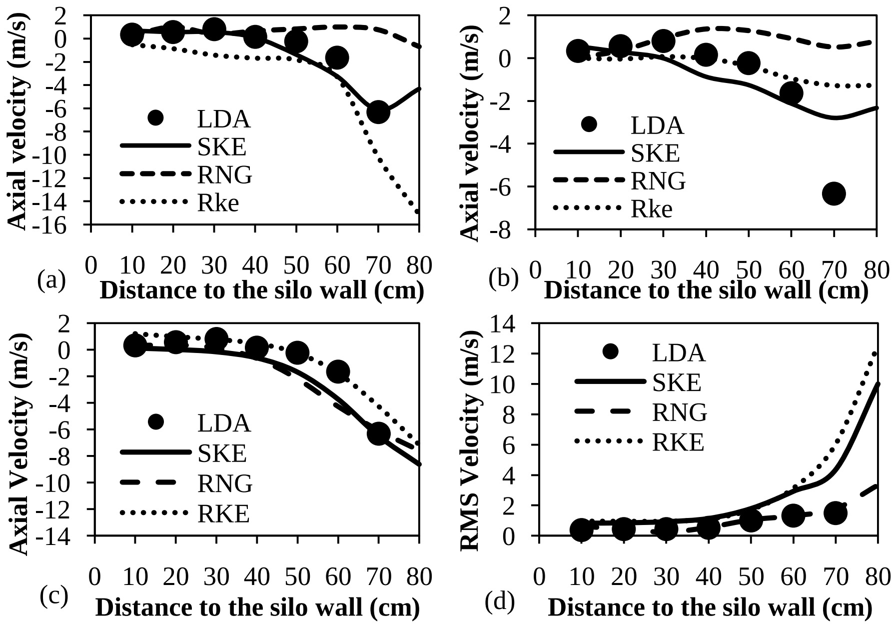

Figure 4. Prediction of (a) axial velocity with standard pressure interpolation and 1st order upwind discretization scheme, (b) axial velocity with PRESTO interpolation and 2nd order upwind discretization scheme, (c) axial velocity with standard pressure interpolation and 2nd order upwind discretization scheme and (d) RMS turbulence with standard pressure interpolation and 2nd order upwind discretization scheme.

References

B.E. Launder and D.B. Spalding, Numerical computation of turbulent flows, Comput. Method Appl., 1974, 3, 269-289.

F. Hauert 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.

G.C. Ke, 2009, Occupational Safety and Health Forensics, The Ingenieur, Vol. 43.

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

T.H Shih, W.W Liou, A. Shabbir, Z. Yang and J. Zhu, 1995, A new k-ε eddy viscosity model for high Reynolds number turbulent flows, Comput Fluids, 24, 227-238.

U. Krause, 2009, Introduction, Fires in Silos: Hazards, Prevention and Fire Fighting. WILEY-VCH Verlag GmbH & Co. KGaA, Weinheim.

V. Yakhot and S.A.Orszag, 1986, Renormalization group analysis of turbulence, I. Basic theory, J Sci Comput., 1, 3-51.