Progress in Nuclear Energy 109 (2018) 204–213

Contents lists available at ScienceDirect

Progress in Nuclear Energy

journal homepage: www.elsevier.com/locate/pnucene

Numerical investigation of jet agitation in a nuclear liquid waste storage tank

V.S. Sanapala[a,b,*], Ritwik Das[a,c], A. Ravisankar[a]

[a] Reprocessing Group, Indira Gandhi Centre for Atomic Research, Department of Atomic Energy, Kalpakkam, 603 102, India
[b] Department of Applied Mechanics, Indian Institute of Technology Madras, Chennai, 600 036, India
[c] The Robotics Institute, Carnegie Mellon University, Pittsburgh, PA, 15213, USA

ARTICLE INFO

Keywords:
Radioactive liquid waste
Particle suspension
Jet agitation
Turbulent eddy viscosity
Storage tank

ABSTRACT

Radioactive liquid waste is often stored in a large capacity (150 m^3) horizontal cylindrical tanks. It is necessary to keep the contents of the tank agitated to prevent the settling of fine solids in the tank. In extreme cases, solids settled at the tank base may invoke the system malfunction. This paper deals with the Computational Fluid Dynamic (CFD) modeling of the agitation mechanism considering the turbulent dispersion effects. The simulations are conducted on a standard tank geometry to arrive the optimum jet velocity required to suspend the settled solid particles thoroughly. The distribution of volume phase fraction, velocity magnitude, turbulence kinetic energy (TKE) and eddy dissipation of each phase for different inlet jet velocities (v = 10–25 m/s) have been presented. The spatial variation of these quantities are measured at three different planes in the vessel mainly in the vicinity of the nozzle exit. The results indicate that the jet velocity is a significant parameter that influences the particle suspension. Parametric studies have also been carried out for four different particle sizes, viz. d_p = 5, 10, 50 and 100 μm. The present study revealed that, for the range of parameters covered, the smallest (d_p = 5 μm) and the largest (d_p = 100 μm) particle sizes has least effect in terms of solids suspension.

1. Introduction

Jet Agitation is a widely been used technique for homogeneous suspension of solid particles in industrial applications such as dissolution, polymerization, chemical synthesis and liquid waste storage tanks (Qi et al. (2013); Natarajan (2017); Wattal (2017)) etc. In these operations, it must be ensured that all the solid particles are kept under vigorous motion in order to prevent them settled at the bottom of the tank (Upson (1984)). Therefore, the radioactive liquid waste stored in the tank may contain fission products and/or fine solid particles must be agitated by multiple jets, which are created by forcing air through nozzles. The distribution of suspended solids in an agitated tank is a function of the field of velocity, characteristics of turbulence and interaction between the solid and liquid phases (Barresi and Baldi (1987)).

In the literature (Jayanti (2001); Zughbi and Rakib (2004); Ochieng and Onyango (2008); Qi et al. (2013); Kong and Fox (2017)), several Computational Fluid Dynamics (CFD) tools are widely been used for evaluating the mixing of fluids and suspension of solids by using jets while understanding the associated transient characteristics. Jayanti (2001) has carried out CFD studies to simulate jet agitation in a cylindrical vessel and the flow circulation patterns and mixing within the vessel. He has studied the effect of various configurations of jets in reducing mixing time. Montante et al. (2001) carried out simulations to investigate the effect of impeller clearance on predicting flow patterns in stirred vessels using sliding grid approach. Micale et al. (2004) have developed a CFD model to find the maximum suspension height of particles in stirred vessels considering only drag force between the fluid and solid particles using the pinneli drag model. Aubin et al. (2004) have investigated the influence of different modeling approaches, discretization methods and selection of turbulence model by comparing the mean velocities, turbulent kinetic energy etc due to stirring in a dished-bottom vessel. Yeoh et al. (2004) have used Large eddy simulation (LES) approach coupled with the sliding/deforming mesh (SDM) to simulate the flow inside a stirred vessel mainly for predicting the dissipation rate of turbulent energy. Rahimi and Parvareh (2005) have studied the effect of different jet angles on the mixing mainly through the comparison of velocity flow patterns using three different turbulence models. Their results indicate that RNG k - ε model is most effective in predicting the flow features such as streamline contours, rotation, and induced vortices etc. Lane et al. (2005) adopted the multi-fluid model implemented in a commercial CFD code for the simulation

* Corresponding author. Reprocessing Group, Indira Gandhi Centre for Atomic Research, Department of Atomic Energy, Kalpakkam, 603 102, India.
E-mail address: svsatyam@gmail.com (V.S. Sanapala).

https://doi.org/10.1016/j.pnucene.2018.08.013
Received 23 March 2018; Received in revised form 19 July 2018; Accepted 16 August 2018
0149-1970/ © 2018 Elsevier Ltd. All rights reserved.

of a gas-liquid stirred vessel by introducing breakage and coalescence terms as user sub-routines functions. Buffo et al. (2012) have simulated a stirred tank reactor using CFD technique coupled with a population balance model (PBM) to solve an Eulerian multi-fluid model are found to be efficient in simulating the polydisperse multiphase flow.

Zughbi and Rakib (2004) have studied effect of different jet angles and the number of jets in reducing the mixing time. Their 3D simulations have proved that the jet injection angle is a significant parameter for improving the mixing. Ochieng and Lewis (2006) have studied the effect of drag force on solids concentration distribution and overall particle suspension in a stirred tank using both CFD simulations and experiments. For low solids concentration (< 1%), the suspension is governed by bulk fluid motion and drag forces whereas turbulent dispersion is significant at high solid concentrations (> 10%) (Ochieng and Lewis (2006)). Ochieng and Onyango (2008) have presented the variation of the particle settling velocity with the particle size and showed the limitation of the Stokes law with larger particle diameters ($d_p < 100$ μm) and for flows of high particle Reynolds number, Re_p (> 1). Fletcher and Brown (2009) have investigated the role of turbulence in developing single-phase flow structures, and the associated solids distribution in stirred tanks. Zadghaffari et al. (2010) have simulated the flow field in a Rushton turbine and compared the numerically estimated mean velocities, turbulence kinetic energy (TKE), turbulent energy dissipation (TED) with corresponding experimental data of Wu and Patterson (1989). They have reported that a good agreement is observed for the measured mean velocity components with those experiments. Yao et al. (2014) have used a coupled CFD-PBM approach to anticipate the hydrodynamic behavior of mutli-phase flow in a 3D fluidized bed reactors. They investigated the effect of drag force between the two phases i. e, gas and solid particles using different drag models, namely Syamlal- O'Brien, Gidaspow, McKeen and energy minimization multi-scale (EMMS) and discussed the merits and demerits of them. Kong and Fox (2017) have developed a solution algorithm that can handle all the flow regimes occurring simultaneously in a fluid-particle system domain. They have introduced the flux-splitting function in order to segregate the free-transport and spatial flux contributions.

The problem of jet agitation for particle suspension is reasonably addressed in the literature. But, most of these studies are confined to either analytical or experimental approach for improving the mixing (or) agitation in stirred vessels. There is a lack of insight information such as flow visualization, distribution of velocity and dissipation of energy etc and detailed characteristics of induced effects with these techniques. To this end, we investigate the dynamics of jet agitation for further understanding of the flow features and turbulence characteristics in suspension of solids by considering two major parameters: jet velocity and particles size. In the present study, the development of a numerical model is discussed and the uniform particle suspension is achieved through the optimization of jet velocities for different particle sizes comparing the distribution of phase fraction, vorticity, TKE, eddy dissipation etc.

2. Governing equations and solution methodology

The governing equations, various models to formulate the interaction and energy exchange between different phases and the approach for solving the problem under investigation are briefly presented in this section.

2.1. Governing equations

The continuity and momentum conservation equations describing the fluid behavior in a two-dimensional unsteady tank for each phase q are as follows:

$$\frac{\partial}{\partial t}(\alpha_q\rho_q) + \nabla\cdot(\alpha_q\rho_q\boldsymbol{u_q}) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\alpha_q\rho_q\boldsymbol{u_q}) + \nabla\cdot(\alpha_q\rho_q\boldsymbol{u_q}\boldsymbol{u_q}) = -\alpha_q\nabla P + \nabla\cdot\overline{\overline{\tau_q}} + \sum_{q=1}^{n}(R_{pq} + m_{pq}\boldsymbol{u_{pq}}) + f_q \tag{2}$$

$$\overline{\overline{\tau_q}} = \alpha_q\mu_q\left(\nabla\boldsymbol{u_q} + \nabla\boldsymbol{u_q}^T\right) + \alpha_q\left(\lambda_q - \frac{2}{3}\mu_q\right)\nabla\cdot\boldsymbol{u_q}\overline{\overline{I}} \tag{3}$$

where, α represents the volume phase fraction, $\boldsymbol{u}$ denotes the velocity vector, ρ is the fluid density, μ is the dynamic viscosity, P is the pressure and f_q refers to the body force. While, $\overline{\overline{\tau_q}}$ is the q^{th} phase stress-strain tensor, $\overline{\overline{I}}$ is the unit tensor and $R_{pq} = K_{pq}(\boldsymbol{u_p} - \boldsymbol{u_q})$, is the interaction force between phases and K_{pq} is the interphase momentum exchange coefficient. Where terms with the subscript pq account for exchange between primary phase p and secondary phase q.

The interphase exchange between the two fluid phases is modelled using Schiller-Neumann model (Schiller and Naumann (1933)), which is acceptable for general use in all fluid-fluid phase flows. The exchange coefficient can be written as $K_{pq} = \frac{3}{4}C_D Re\frac{\alpha_q\alpha_p\mu_q}{d_p^2}$, where C_D is the drag coefficient, d_p is the diameter of droplet or bubble and Re is the relative Reynolds number, $Re = \frac{\rho_q\left|u_p - u_q\right|d_p}{\mu_q}$. In order to describe the effects of turbulent velocity fluctuations and other scalar quantities in multiphase flows the standard k - ε model is deployed in the present study. The k and ε equations representing this model are as follows:

$$\frac{\partial}{\partial t}(\alpha_q\rho_q k_q) + \nabla\cdot(\alpha_q\rho_q U_q k_q) = \nabla\cdot\left(\alpha_q\frac{\mu_{t,q}}{\sigma_k}\nabla k_q\right) + \alpha_q G_{k,q} - \alpha_q\rho_q\varepsilon_q \tag{4}$$

$$\frac{\partial}{\partial t}(\alpha_q\rho_q\varepsilon_q) + \nabla\cdot(\alpha_q\rho_q U_q\varepsilon_q) = \nabla\cdot\left(\alpha_q\frac{\mu_{t,q}}{\sigma_\varepsilon}\nabla\varepsilon_q\right) + \alpha_q\frac{\varepsilon_q}{k_q}(C_{1\varepsilon}G_{k,q} - C_{2\varepsilon}\rho_q\varepsilon_q) \tag{5}$$

$G_{k,q}$ is the production of turbulent kinetic energy ε_q is the dissipation rate and U_q is the phase-weighted velocity. $\mu_{t,q} = \rho_q C_\mu\frac{k_q^2}{\varepsilon_q}$. Here, $\mu_{t,q}$ is the turbulent viscosity in terms of the turbulent kinetic energy of phase q. Where C_μ, $C_{1\varepsilon}$, $C_{2\varepsilon}$, σ_k and σ_ε are the standard parameters used in the k - ε model and the values are given as $C_\mu = 0.09$, $C_{1\varepsilon} = 1.45$, $C_{2\varepsilon} = 1.9$, $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.3$.

Similarly, the interaction between solid and fluid phases pq is replaced by fs to arrive the following correlation given by Gidaspow (Gidaspow et al. (1991))

$$K_{fs} = \begin{cases} \frac{3}{4}C_D\frac{\alpha_s\alpha_f\rho_f\left|u_f - u_s\right|}{d_s}\alpha_f^{-2.65}, & \alpha_f > 0.8 \\ 150\frac{\alpha_s(1-\alpha_f)\mu_f}{\alpha_f d_s^2} + 1.75\frac{\alpha_s\rho_f\left|u_f - u_s\right|}{d_s}, & \alpha_f \le 0.8 \end{cases} \tag{6}$$

$$C_D = \frac{24}{Re\alpha_f}(1 + 0.15(Re\alpha_f)^{0.687}) \tag{7}$$

Where, $Re = \frac{\rho_f\left|u_f - u_s\right|d_s}{\mu_f}$. The effect of turbulent dispersion due to instantaneous fluctuations is accounted through a correction term to inter-phase momentum equation.

The solid is modelled as a pseudo fluid and the pressure and stress tensor terms are given by kinetic theory of granular flow (Ding and Gidaspow (1990)). Assuming that the granular fluctuation energy of particles is in local equilibrium and the convection and diffusive terms are ignored, the transport equation for granular temperature, Θ_s, is given as (Hosseini et al. (2013))

Download English Version:

https://daneshyari.com/en/article/11007425

Download Persian Version:

https://daneshyari.com/article/11007425

Daneshyari.com