Applied Mechanics and Materials Vol. 372 (2013) pp 340-345
© (2013) Trans Tech Publications, Switzerland
doi:10.4028/www.scientific.net/AMM.372.340

Flow analysis of melted urea in a perforated rotating bucket

Aadil Muhammad, Nejat Rahmanian and Rajashekhar Pendyala

Chemical Engineering Department, Universiti Teknologi PETRONAS, Bandar Seri Iskandar, 31750 Tronoh, Perak, Malaysia.

The Corresponding Author: nejatollah.r@petronas.com.my; rahmaniyan@yahoo.com

Keywords: CFD, Prilling, Perforated rotating bucket, Urea

Abstract. A comprehensive study of the internal flow field for the prilling application in a perforated rotating bucket has been carried out. Computational Fluid Dynamics (CFD) is used to investigate the flow field of urea melt inside the perforated rotating bucket. The bucket is mounted at the top of the prilling tower. In prilling process, urea melt is sprayed by the perforated rotating bucket to produce the urea droplets, which falls down due to gravity. These drops fall down through a cooling medium and solidify into prills. The velocity field in the bucket is very important to study, as it has great effect on the heat and mass transfer performance in prilling process. ANSYS 14.0 CFD package is used to simulate and Design Modeler and Catia V5 are used for geometrical model of the perforated prilling bucket. Velocity distribution on different planes are obtained and discussed.

Introduction

Prilling process is a significant field of study in fertilizer industry due to high demand of fertilizer especially for agriculture products such as urea. A large number of chemical products are produces by this process. Prilling is a technique of producing reasonably uniform spherical particles from urea melt. Prilling process is considered as the finishing step in urea production plant [1]. Urea melt is pumped into the perforated rotating bucket which is installed at the top of prilling tower. This bucket is rotated at high speed and urea melt flows out from the orifices present on the wall of the bucket. In prilling process, the appearance, strength, flow ability, mechanical and physical properties of desired solid prills are very important, and can be improved by manipulating the parameters such as rotating speed, viscosity of molten fluid and orifice sizes [2]. Prilling is also characterized as a particular type of granulation. It is widely used in the fertilizer, food, ceramics, catalysts, ANFO and especially in pharmaceuticals industries [3,4]..

Urea and other fertilizers produced by prilling process are at lower cost as compared to the granulation process . Granulation process is more complex associated with large investment and high operating cost. In prilling , urea melt is sprayed to produce the drops falling through a cooling medium that are crystallized into particles of spherical forms under the influence of surface tension. The main technology in the prilling is to develop the spray via perforated rotating bucket. Prilling essentially consist of two steps. Firstly, producing the liquid droplets and secondly solidifying them by the cooling through a rising ambient air stream. The most suitable materials for prilling are those who melt without decomposition and preferably have high melting point to permit the use of ambient air for cooling. The melt should have a very low viscosity, but a high surface tension at temperatures just a few degrees above the melting point. Despite of wide interest and huge investigation, various aspects of the behavior of fluid flow in perforated rotating bucket are still not well understand. Prilling technology is very successful but still limited and very little research has been published in literature. With the recent work in measurement methods, the internal details in the bucket start to be exposed. Various studies which have influence on the bucket have been attempted [5,6]. A few companies are doing research to optimize the prilling operation, such as Stamicarbon (Netherlands), Snamprogetti (Italy), and Mistui (Japan).

The ambition of this work is to provide a vision of CFD application to understand the flow field of urea melt in the perforated rotating bucket. This work is extension of previous work done for the single orifice to check the behavior of flow regime [7]. The unique features associated in CFD has been used to study the fluid flow of urea melt.

Materials and Methods

There are significant attempts to model and predict the behavior of flow structure in perforated rotating bucket. Urea melt is fed from the top of rotating bucket at its melting point of 132^{0}C. Urea melt is used as a model material. The process is affected by the physical and chemical properties of the feed material, process parameters and model of the bucket. Therefore, the model can be based on a number of parameters such as rotation speed, urea melt properties, arrangement and the size of orifices. The geometrical model has been designed. Catia (V5R17) software package is employed to design the model accordingly as shown in Figure 1.

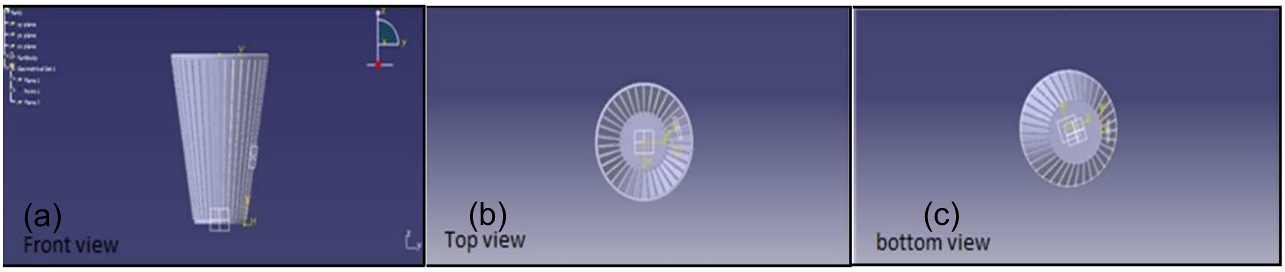

Fig. 1 Geometrical model of perforated rotating bucket in 3D,
(a) front view, (b) top view,(c) bottom view

The dimensions for the bucket geometry are set according to Friestad data [8], which is given in Table 1. As the bucket is rotating at high speed, due to centrifugal force urea melt in the form of droplets falls out from perforated the bucket wall. Then this geometry is subdivided into computational domain. Inlet for the urea melt is introduced at the top of the bucket is 0.177 kg/sec, The outlet is from orifices present at bucket wall at atmospheric pressure and the bucket is rotating at its vertical axis at the speed of 630 rpm.

Table 1:Dimensions for the perforated prilling bucket

Parameters	Values
Upper diameter for prilling bucket [mm]	150
Lower diameter for prilling bucket [mm]	80
Height [mm]	200
Orifice diameter for top row [mm]	3.8
Orifice diameter for bottom row [mm]	2.4
Number of orifice rows [-]	64
Orifices per row [-]	35
Total no of orifices [-]	2240

Numerical Framework

ANSYS 14.0 software package provides a list of criteria to assess the quality of mesh through mesh associated parameters such as the element ratio, connectivity number and maximum and minimum face angle. The model is then discredited into number of small volumes using mesh scheme. The mesh size is of great importance and have to select carefully for the accuracy of simulation results. To determine the sensitivity of results three different meshes have been used. A recommended approach to eliminate the influence of mesh size is to seek mesh independent solutions by testing with gradually reduced mesh sizes until the simulation results no longer change [9-11].

Turbulence modeling plays an important role in many of engineering applications and simulation has gone intensive research throughout the years. In order to develop a safe process it is necessary to predict the accurate turbulent model. Turbulence models are generally categorized according to the governing equations they used, such as Reynolds-averaged Navier-Stokes or Large Eddy Simulation equations. These models are also further categorized by the number of additional transport equations which are solved in order to compute the model assistance. Two equation models make a bulk of turbulence models used for CFD simulations. Of all turbulence models available, the standard turbulence model κ-ε still remain an industrial standard and its successful applications are found in the literature [12,13]. This model is based on the solution of equations for the turbulent kinetic energy, K, and the turbulent dissipation rate ε [14,15]. In the standard κ-ε model, the turbulent kinetic energy, κ; and its rate of dissipation, ε, are obtained from the following transport equations:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\mu_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon_k \tag{1}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t}+\frac{\partial(\rho \varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\mu_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_1\frac{\varepsilon}{k}(G_\varepsilon+C_3G_b)-C_2\rho\frac{\varepsilon^2}{k_\varepsilon} \tag{2}$$

The turbulent viscosity is defined as

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

where G_k represents the generation of turbulence kinetic energy due to mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, C_1, C_2, C_3 and C_μ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for κ and ε.

Results and Discussion

The results of CFD simulation shows typical fluid dynamics prediction in the perforated rotating bucket, where the fluid is forced to move the orifices at the bucket wall. The slope of the bucket all converts the centrifugal momentum into vertical direction and create an upward flow near the wall. A simple solution method and the first order scheme of pressure momentum is applied for the perforated rotating bucket. In this analysis, three different grid systems have been used to discover the variation in the results but it follows the same trend for all grid system. This indicates the validity and independency of the model used.

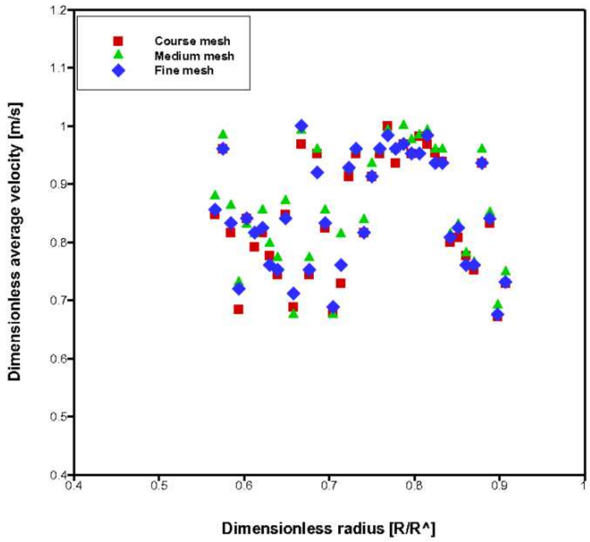

Fig. 2 Quantitative comparison of dimensionless average velocities with various meshes

Quantitative comparison of average velocity with different mesh system is shown in Figure 2. It shows clearly that the average velocities at various orifice positions are similar for all the mesh cases and the tested meshes have similar performance in simulating the model.

Velocity profiles at each plane is analyzed by CFD simulation. Average velocity is also calculated at all 46 planes. Velocity field at four planes are shown in Figure 3. Figure 3 show the velocity fields on horizontal planes at (a) 12 mm, (b) 72 mm, (c) 112 mm and (d) 184 mm respectively. All the planes shows complex velocity field with high rotation speed effect. Figure 3 shows the fluid is rotating and travelling towards the bucket wall orifice. Figure 4 presents the average velocity at each orifice position. A good prediction of the average velocity is seen at every orifice position.

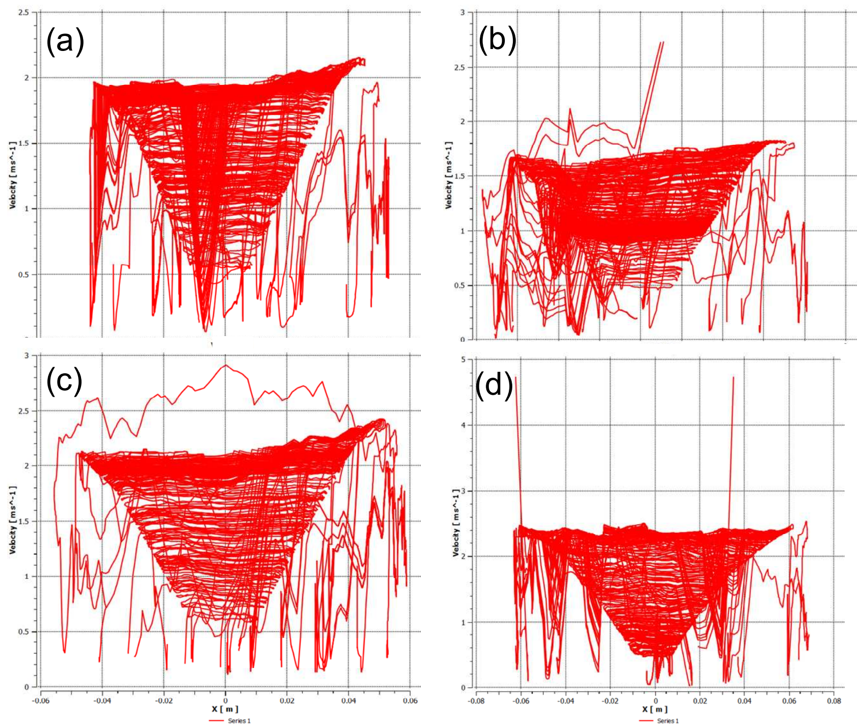

Fig. 3 Velocity profile on different planes along horizontal axis

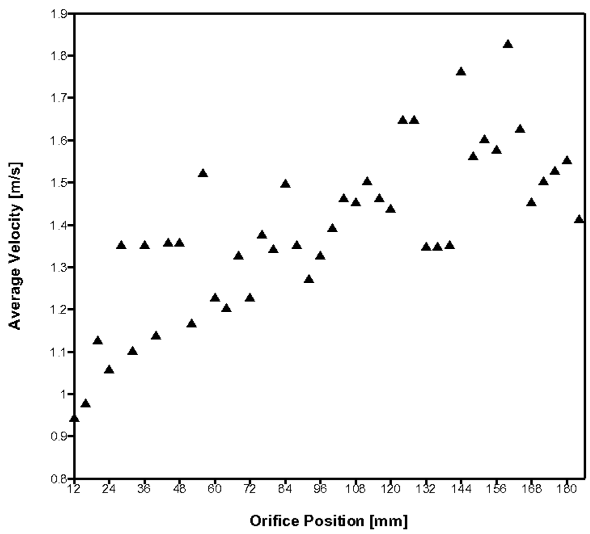

Fig. 4 The average velocity at each orifice position by CFD simulation

Conclusions

In commercial application of perforated rotating bucket, velocity profile is very important for the design aspects or during operations. CFD approach was applied to simulate the flow behaviour inside the perforated rotating bucket. The standard κ-ε model is used for the turbulent flow. The results show complex velocity fields. The inspection of CFD model shows the high fraction of urea melt close to the bucket wall region. The average velocity at outlet orifices ranges from 1 to 1.7 m/s. Circular flow of urea is formed at the centre of the bucket.

Acknowledgement

Dr N. Rahmanian would like to thank Universiti Teknologi PETRONAS for the financial support (Grant No.: STIRF 31/2011) and Ministry of Science, Technology and Innovation (Grant No: 06-02-02-SF0178), Malaysia for their financial support.

Reference

[1] A. Alamdari, A. Jahanmiri, and N. Rahmaniyan, Mathematical modelling of the urea prilling process, Chem. Eng. Comm., 178, (2000) 185-198.

[2] N. Rahmaniyan, A. Alamdari, A. Jahanmrir, Modeling of the urea prilling tower, 3rd World Congress on Particle Technology, Brighton, UK (1997) 94-100.

[3] N. Rahmanian, M. Homavoonfard, and A. Alamdari, Urea prilling process: Co-current vs Counter-current. International Conference on Process, Enviormental and Material Engineering (ICPEAM 2012), Kuala Lumpur, Malaysia.

[4] N. Rahmanian, M. Homayoonfard, A. Alamdari, Simulation of urea prilling process: An industrial case study. Chemical Engineering Communication 200 (6): (2013) 1-19.

[5] A. M. Nilpawar, G. K. Reynolds, A. D. Salman, M. J. Hounslow, Velocity distribution in high shear granulator. In: World Congress of Chemical Engineering, Glasgow (2005).

[6] J. Bridgwater, D. F. Bagster, S. F. Chen, J. H. Hallam, Geometric and dynamic similarity in particle mixing. Powder Technology 2, (1968) 198–206.

[7] M. Aadil, N. Rahmanian, P. Rajashekhar, Analysis of fluid flow of urea in a perforated rotating bucket: Single orifice case. Journal of Applied Sciences, accepted (2013).

[8] I. A. Friestad, Means for feeding fluid materials to a prilling bucket. United States Patent 3900164 (1975).

[9] H. K. Versteeg and W. Malalasekera, An introduction to Computational Fluid Dynamics: The finite volume method. Prentice Hall. 2nd Ed (2007).

[10] H. A. Olvera and A. R. Choudhuri, Numerical simulation of hydrogen dispersion in the vicinity of a cubical building in stable stratified atmospheres. Int. J. Hydrogen Energy., 31: (2006) p. 2356-2369.

[11] Release 11.0 Documentation for ANSYS Workbench, ANSYS Ltd., (2006).

[12] A. M. Foster, R. Barrett, S. J. James, and M. J. Swain, Measurement and prediction of air movement through doorways in refrigerated rooms. International Journal of Refrigeration, 25, (2002) 1102-1109.

[13] A. M. Foster, M. Madge, and J. A. Evans, The use of CFD to improve the performance of a chilled multi-deck retail display cabinet. International Journal of Refrigeration, 28, (2005) 698-705.

[14] W. P. Jones, B. E. Launder, The prediction of laminarization with a two-equation model of turbulence. Int. Journal of Heat and Mass Transfer, 15 (1971).pp. 301-314.

[15] W. P. Jones, B. E. Launder, The prediction of low-Reynolds-number phenomena with a two-equation model of turbulence. Int. Journal of Heat and Mass Transfer, 16 (1973) pp. 1119-1130.

[16] V. Yakhot, S. A. Orszag, Renormalization group analysis of turbulence: I. Basic theory. Journal of Scientific Computing 1 (1), (1986) 1–51.