Transient numerical simulation for solid-liquid flow in a centrifugal pump by DEM-CFD coupling

Si Huang*, Xianghui Su and Guangqi Qiu

School of Mechanical and Automotive Engineering, South China University of Technology, Guangzhou, China

(Received 28 September 2014; final version received 17 April 2015)

The two-phase solid-liquid flow in the computational fluid dynamic (CFD) method is generally analyzed by treating the solid particles as a quasi-fluid element, but this does not take into account the effects of the physical features of the solid particles and collisions among the particles. In this study, the discrete element method (DEM) was coupled with the CFD method in order to analyze the transient two-phase solid-liquid flow in a single-stage centrifugal pump with consideration for the particle-particle and particle-structure interactions. The numerical simulation process involving DEM and FLUENT is described. Simulations were performed for the water at 25°C at the continuous phase together with 15% of the volume as spherical solid particles (density \( \rho = 1500 \text{ kg/m}^3 \)) with the diameter ranging from 1 to 3 mm in a random order. The results illustrate the motion of the solid particles in the pump and their effect on the flow performance of the pump in terms of time variation in the head. The velocity fields for the two-phase flow together with the volume fraction distributions and trajectories of the solid particles in the centrifugal pump are also presented.

Keywords: DEM-CFD coupling; centrifugal pump; trajectories of particles; transient two-phase flow

1. Introduction

Centrifugal pumps are widely used in various slurry transport systems and are considered to be more cost effective than transportation by truck or conveyor belt. Transportation of slurry mixture via centrifugal pumps is most commonly found in operations to dredge various bodies of water, maintain navigation in harbors and rivers, alter coastlines, and collect material for landfill and construction purposes. As well as the above, the mining, fertilizer and food-processing sectors also employ large-scale slurry transportation via pumps. Moreover, preservation of the environment requires that waste be conveyed to a disposal site via pumps. The operational efficiency of centrifugal pumps employed in slurry transportation strongly depends on the pump design and the characteristics of two-phase slurry flow (Addie, Roudnev, & Sellgren, 2007; Clark & Abbott, 1992; Lyczkowski & Bouillard, 2002; Messa & Malavasi, 2014; Wilson, Addie, Sellgren, & Clift, 2006). A systematic study of two-phase solid-liquid flow in pumps and the effects of solid particle size on flow characteristics is thus essential in order to seek crucial design guidance for enhancing the operational efficiency of the transportation process.

The numerical analyses of two-phase flow in various applications have mostly been conducted using the computational fluid dynamic (CFD) method, sometimes coupled with the discrete element method (DEM). In the CFD method, the solid particles in the mixed flow are typically treated as a pseudo-liquid medium in order to obtain the particles’ motion through solutions of the governing equations of multiphase flow models. This method has been widely used in a range of engineering applications due to its global accuracy in two-phase flow prediction, its applicability to a wide range of solid volume fractions and its relatively low computational demand. Simulations of multiphase flow models can be conveniently carried out using commercially available CFD code. This is evident from the various reported studies on two-phase solid-liquid flow in different rotating machinery (Cheng, Li, Gao, & Guo, 2013; Chochua, Koch, & Sorokes, 2005; Fang, Ling, & Sang, 2013; Han, Ma, Li, & Li, 2012; Hua, Yi, & Yu-Liang, 2013; Ma, Wu, Huang, Yu, & Wang, 2013; Pagalthivarthi, Gupta, Tyagi, & Ravi, 2011; Roudnev, Bourgeois, & Kosmicki, 2009; Roudnev & Kosmicki, 2007; Zhang, Li, Cui, Zhu, & Dou, 2013), which have substantially contributed to the design of more efficient rotating machines.

The CFD method, however, exhibits inherent limitations due to the pseudo-fluid approximation of the solid particles, primarily due to discontinuities caused by the motion of the solid particles. The effects of various particle features such as material properties, shape and size cannot be accurately evaluated using the CFD method. Moreover, the CFD method does not provide an accurate assessment...
fluid phase flow is initially analyzed using CFD simulations through solutions of the transient Reynolds-averaged Navier-Stokes (RANS) equation and the standard k-epsilon turbulence model in the FLUENT platform. The flow simulation is iterated until convergence is achieved for a given time step. The DEM simulations are subsequently initiated to compute the instantaneous drag force acting on each DEM particle from the transient fluid conditions of the mesh cell containing the particular particle. The simulations are continued using the DEM to determine the instantaneous position, velocity and forces developed by the particle. The coordinates of the particles are subsequently updated and transferred to the CFD solver, together with the inertia forces due to the particles. For this purpose a momentum source is introduced to each mesh cell to represent the effect of energy transfer from each DEM particle, taking into account the volume fraction of the CFD mesh cell occupied by the solid particle.

In the DEM-CFD coupling process, the DEM iterations are performed in an independent integration time step following the convergence corresponding to each CFD simulation time step. The total simulation time is thus significantly higher, especially when the dynamic motion of a large number of solid particles is tracked. The total simulation time for a single iteration, however, is affected not only by the number of particle elements but also by the surface and geometric properties of the elements, and the contact conditions. EDEM simulations are nearly linearly dependent on the number of elements.

2. Material and interaction data

The simulations were performed for water at 25°C as the continuous phase, while the solid particles formed the discrete phase. The physical parameters of the solid particles were chosen so as to approximately simulate the conditions encountered in most slurry transport applications (Wilson et al., 2006). The density of the particles was taken as 1500 kg/m³, while the sizes of the particles were varied from 1.0 to 3.0 mm in diameter in a random manner. The volume fraction of the solid particles was set to 15% at the pump inlet. Some of the material properties of the particles

---

**Figure 1. EDEM-FLUENT coupling process.**

---
Table 1. Material properties of the solid particles and the pump structure.

| Material | Poisson’s ratio | Shear modulus (MPa) | Density (kg/m³) | Diameter (mm) | Volume fraction at pump inlet (%) |
|----------|----------------|---------------------|-----------------|---------------|----------------------------------|
| Pump     | 0.30           | 70.0                | 7800            | –             | –                                |
| Particles| 0.40           | 21.3                | 1500            | 1.0–3.0       | 15                               |

Table 2. Model parameters for the particle-particle and particle-pump interactions.

| Interaction           | Coefficient of restitution | Coefficient of static friction | Coefficient of rolling friction |
|-----------------------|----------------------------|-------------------------------|--------------------------------|
| Particle-particle     | 0.44                       | 0.27                          | 0.01                           |
| Particle-pump         | 0.50                       | 0.15                          | 0.01                           |

2.3. Computational domain and meshing

The computational domain consisted of the inlet pipe, impeller, volute and outline pipe of a centrifugal pump, as shown in Figure 2. A single stage centrifugal pump was selected for the study of two-phase flow and the interactions of the solid particles. The flow rate capacity of the pump was 88.5 m³/h with the head being 14.5 m, when operating at 1450 rpm. The impeller comprised a total of six blades with a wrap angle of 100° (inlet diameter $D_0 = 125$ mm; outlet width $b_2 = 26.5$ mm). The hexahedral meshes were created using Integrated Computer Engineering and Manufacturing code (ICEM) and a total of 1,019,538 cells constituted the entire computational domain. Sliding meshes were adopted at the interfaces of the rotating impeller with the stationary components of the pump for the transient analyses. In addition, the acceleration due to gravity ($g = 9.81$ m/s²) was taken into account and its direction was considered to be opposite to the flow entering the pump inlet. This type of inlet-downward installation is quite often found in slurry pumps used for dredging operations in rivers.

2.4. Transient analysis

Both the pump and the fluid medium were considered to be in a static state prior to start-up. Once the pump starts, the impeller rotational speed grows with time until the design value and the flow field inside the pump gradually enter a stable circulating state (Saito, 1982; Tsukamoto & Ohashi, 1982). The simulations were designed to study the transient solid-liquid flow fields but without factoring in the contributions due to the transient response of the impeller, in order to preserve reasonable computational efficiency. A steady impeller speed of 1450 rpm was thus considered in the transient analysis. The initial velocity of the liquid phase was set as zero in all computational domains. Computations were initiated with solid particles being released continuously at the pump inlet, while the volume of solid particles was maintained at 15%. The inlet boundary condition in FLUENT included the design flow rate of the pump, while the outlet boundary was given by the outlet pressure ($p = 10^5$ Pa). The computations in FLUENT were performed using the time step $\Delta t = 60/65nZ \approx 10^{-4}$ s, which represents an impeller rotation of less than one degree. A considerably smaller integration time step, however, is essential for simulations in EDEM software for capturing the contact behavior of the solid particles accurately (DEM, 2010). The time step of $10^{-6}$ s was thus selected for the EDEM simulations and the ratio of DEM-CFD simulation steps was thus 1:100. The absolute convergence criteria for the calculated residuals were set as $10^{-3}$ by default, which was monitored in terms of the pump head (total pressure difference between the outlet and the inlet) during the computations. The computations were terminated on the basis of the above convergence condition together with the regular variations in the pump head.
3. Results and discussion

3.1. Time-dependent performance of the solid-liquid pump

The computations were performed with the pump design condition taken into account (flowrate, \(Q_l = 88.5 \text{ m}^3/\text{h};\) impeller speed, \(n = 1450 \text{ rpm}\)). The transient response of the pump was obtained in terms of the head using the FLUENT post-processor. Figure 3 compares the variation in the pump head with time for the two-phase solid-liquid flow and the water-only flow. The results show periodic oscillations in the pump head for both flow conditions. The amplitudes of oscillations in both cases approach nearly steady values as \(t\) exceeds three times the fundamental rotation period of the impeller (\(T = 0.0414 \text{ s}\)). It should be noted that the settling time of \(3T\) for the pump head corresponds to the conditions considered, namely, constant impeller speed and zero initial velocity of the liquid phase in all domains, as stated in section 2.4 above.

The results further show oscillations in the pump head at a frequency of 145 Hz for both the flows, which directly relates to the product of impeller blades and the fundamental frequency of the impeller (24.16 Hz). Within a rotation period \(T\), there are six peak values of \(H\), corresponding to the impeller vane number. The amplitude of oscillations in the pump head for two-phase solid-liquid flow, however, is considerably higher than that for water alone, while the steady-state mean value of the head for the purely liquid phase flow is higher than that for the two-phase flow.

Figure 4 shows the variations in the normalized solid particle volume \(V'_p\) with time. The results also suggest that the particles entering the pump are more than those discharged during the period. Small amplitude variations in the volume of normalized particles were also observed during the interval \(0.28 \leq t \leq 0.35 \text{ s}\), which is likely due to the clustering of the solid particles during discharge.

3.2. Trajectories of the solid particles

Figure 5 shows the trajectories of the solid particles inside the pump at various instants during the simulation. The computational domains of the inlet and the outlet pipes are omitted so as to clearly observe the distributions of the solid particles phase around the impeller and the volute. The trajectories clearly show a gradually increasing particle volume with time for \(t \leq 0.3 \text{ s}\), as seen in Figure 4. The solid particles tend to distribute more uniformly in the inlet section (Figure 5(a)), and enter the impeller along the pressure sides of the vanes (Figure 5(b–e)). In general, the particles tend to maintain a steady trajectory during flow towards the volute that corresponds with the shape of the impeller vanes, as seen in Figure 5(c). The particles tend to cluster along the volute outside wall and move downstream towards the outlet (Figure 5(d)).

Figure 6 further shows the detailed particle trajectories around some of the impeller vanes, which is an enlarged view of the particles’ motion shown in Figure 5(f). This enlarged view illustrates the motion of particles of different sizes — the trajectories of different-sized particles could not be distinguished from the figure. This is due to frequent particle-particle and particle-pump interactions. The large particles are presented in warmer colors, while smaller particles are shown in cooler colors. The trajectories further show that the solid particles dominantly move to the pressure sides of the impeller vanes. This tendency has also been observed in an experimental study of flow in a similar specific speed centrifugal pump (Liu, Xu, & Tang, 2008). The particles, however, tend to deviate from the pressure...
Figure 5. Trajectories of the solid particles inside the pump at different selected instants.

Figure 6. Trajectories of solid particles inside the pump ($t = 0.35$ s).

Figure 7. Variations in the volume-average velocities of the liquid and solid phases in the flow domains.
Figure 8. Velocity fields of the two phases of flow inside the pump (left-hand side: liquid-phase velocity; right-hand side: solid-phase velocity).
sides near about one third of the length of the vanes while moving downstream.

3.3. **Velocity fields of the flow phases**

Figure 7 shows the variations in the volume-average velocities of both flow phases in the pump domains, which are obtained from the EDEM-FLUENT post-processors. The results show that the average velocity of the liquid phase approaches a nearly steady value of 5.7 m/s, while that of the solid phase approaches a considerably lower value of 4.7 m/s. There are apparently whole slip velocities between the liquid and solid phase flows inside the pump.

Figure 8 shows the velocity fields of the liquid phase (left side) and the particle phase (right side) inside the pump at different selected instants. The figures show quite similar velocity fields for the liquid phase at all of the selected instants. This is due to the fact that the volume-average velocity of the liquid phase in the pump domain rapidly approaches a steady value after \( t = 0.025 \) s, as seen in Figure 7. The flow velocity magnitude near the impeller outlet tends to be relatively high due to the rotating impeller, while the maximum was obtained near 16.4 m/s. In contrast, the velocities of the solid particles are substantially lower, particularly at the inlet. The particle velocity is only about 2 m/s in the inlet pipeline, while they tend to accelerate in the impeller and discharge from the impeller outlet at a speed of about 10 m/s. From Figure 7, it is evident that the volume-average velocity of the solid particle phase in the pump domains approaches a steady value after \( t = 0.08 \) s. The velocity fields for the particle phase at the instants \( t = 0.10 \) s and \( t = 0.30 \) s thus appear to be quite similar. While the higher velocity of the liquid phase compared to that of the particle phase in the inlet pipeline and the impeller is clearly evident, the velocity slips between both the phases are not obvious in the volute. This is due to the fact that the liquid-phase velocity decreases considerably with increase in the cross-sectional area of the volute, while the solid particles, as the dispersed phase, retain their inertia from the impeller.

3.4. **Solid-phase volume distribution**

The distributions of the particle volume fraction, as well as the trajectories of the solid particles inside the pump, can provide important guidelines for identifying potential locations of wear caused by the solid particles. The contours of the particle volume fraction \( \alpha \) inside the pump are thus evaluated from the coupled DEM-CFD simulations and presented in Figure 9. The simulation results revealed a maximum particle volume fraction \( \alpha_{\text{max}} \) in the equilibrium state \( (t \geq 0.30 \text{ s}) \) in the order of 0.5 occurring on the outside wall of the volute. More particles tend to cluster together on the inlet-downward side of the volute wall due to the gravity effect, especially in the downstream path of the volute, since the kinetic energy of the two-phase flow decreases gradually. The rate of erosion of the pump wall due to the particles \( E_r \) can be calculated as follows (DEM, 2010):

\[
E_r = E\dot{N}m_p, \tag{1}
\]

where \( m_p \) is the mass of the particle, \( \dot{N} \) is particle number rate and \( E \) is a constant, which is a function of the particle impact velocity, the impact angle, and the particle and wall properties. The software offers two erosion models for computing the constant \( E \), namely, the models reported by Finnie (1960) and Grant and Tabakoff (1975). Further studies of the erosion of the pump components due to interactions with solid particles would be desirable using the results obtained from coupled DEM-CFD simulations accounting for the effects of particle size and volume fractions.

4. **Conclusions**

The transient two-phase solid-liquid flow in a centrifugal pump was analyzed using the coupled DEM-CFD simulations. The conclusions are as follows:

1. For the two-phase flow with densely distributed particles (1500 kg/m\(^3\) in density, 1.0–3.0 mm in diameter, 15% volume fraction at the pump inlet), the results show that the pump head in the equilibrium state is considerably lower compared to that for the liquid-only flow. The two-phase flow also results in substantially higher amplitude oscillations in the pump head, an effect which is attributed to collisions among the particles in the densely distributed state. The frequency of the pump head oscillations, however, is identical for both flows,
which could be directly related to the fundamental frequency of the impeller and the number of impeller vanes.

(2) The results further show an increase in the total volume of the particles inside the pump during the start-up period, suggesting that the particles entering the pump are greater than those discharged in the same period. The velocity fields of the flows also reveal whole slip velocities between the two phases inside the pump.

(3) The solid particles are more uniformly distributed in the inlet section, and enter the impeller along the pressure sides of the vanes. The trajectories of the particles are consistent with the shape of the impeller vanes when moving towards the volute. Furthermore, most of the particles along the outside wall of the volute move to the downstream towards the outlet. The effect of particle size on the trajectory, however, could not be distinguished due to frequent particle-particle interactions and higher concentration of the particles. The results also suggest that the contribution due to the effect of gravity cannot be ignored in the analysis.

(4) The results in this study are limited to the solid, liquid and two-phase flows in a pump; it would be desirable to compare these with those obtained from the pseudo-fluid multiphase flow approach so as to assess the relative errors. Further studies on the applications of the results for predicting pump component wear due to interactions with solid particles are also desirable.

Disclosure statement
No potential conflict of interest was reported by the authors.

References
Addie, G. R., Roudnev, A. S., & Sellgren, A. (2007). The new ANSI/HI centrifugal slurry pump standard. Journal of the South African Institute of Mining and Metallurgy, 107(6), 403–409.
Cheng, X. R., Li, R. N., Gao, Y., & Guo, W. L. (2013). Numerical research on the effects of impeller pump-out vanes on axial force in a solid-liquid screw centrifugal pump. IOP conference series: Materials science and engineering. The 6th international conference on pumps and fans with compressors and wind turbines (ICPF2013), September 19–22, Tsinghua University, Beijing, China, Paper No. 062008.
Chochua, G., Koch, J. M., & Sorokos, J. M. (2005). Analytical and computational study of radial loads in volutes and collectors. American Society of Mechanical Engineers. Reno, Nevada, USA, 871–879.
Clark, D., & Abbott, C. (1992). Improve slurry pump life. Hydrocarbon Processing, 71(7), 53–55.
Cundall, P. A. (1971). A computer model for simulating progressive, large scale movements in blocky rock systems. Proceedings of the international symposium on rock mechanics. Nancy France, 8–12.
Fang, J., Ling, X., & Sang, Z. F. (2013). Solid suspension in stirred tank equipped with multi-side-entering agitators. Engineering Applications of Computational Fluid Mechanics, 7(2), 282–294.
Finnie, L. (1960). Erosion of surfaces by solid particles. Wear, 3(2), 87–103.
FLUENT (16.0) [Computer software]. http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/Fluid+Dynamics+Products/ANSYS+Fluent.
Grant, G., & Tabakoff, W. (1975). Erosion prediction in turbomachinery resulting from environmental solid particles. Journal of Aircraft, 12(5), 471–478.
Han, W., Ma, W., Li, R., & Li, Q. (2012). The Numerical analysis of radial thrust and axial thrust in the screw centrifugal pump. Procedia Engineering, 31, 176–181.
Hua, T., Yi, L., & Yu-Liang, Z. (2013). Numerical analysis of a prototype centrifugal pump delivering solid-liquid two-phase flow. Journal of Applied Sciences, 13(7), 3416–3420.
Kafui, K. D., Thornton, C., & Adams, M. J. (2002). Discrete particle-continuum fluid modelling of gas-solid fluidized beds. Chemical Engineering Science, 57(13), 2395–2410.
Liu, J., Xu, H., & Tang, S. (2008). Experimental research on the movement rule of solid particles in centrifugal pump. Journal of Hydroelectric Engineering, 27(6), 168–173.
Lyczkowski, R. W., & Bouillard, J. X. (2002). State-of-the-art review of erosion modeling in fluid/solids systems. Progress in Energy and Combustion Science, 28(6), 543–602.
Ma, X. D., Wu, D. Z., Huang, D. S., Yu, H., & Wang, L. Q. (2013). CFD analysis on a turbulence generator of medium consistency pump. IOP conference series: Materials science and engineering. The 6th international conference on pumps and fans with compressors and wind turbines (ICPF2013), September 19–22, Tsinghua University, Beijing, China, Paper No. 032003.
Messa, G. V., & Malavasi, S. (2014). Numerical prediction of particle distribution of solid-liquid slurries in straight pipes and bends. Engineering Applications of Computational Fluid Mechanics, 8(3), 356–372.
Pagalithvarthi, K. V., Gupta, P. K., Tyagi, V., & Ravi, M. R. (2011). CFD predictions of dense slurry flow in centrifugal pump casings. World Academy of Science, Engineering and Technology, 5(3), 16–28.
Roudnev, A. S., Bourgeois, R. J., & Kosmicki, R. J. (2009). Slurry pump casing wear prediction using numerical multiphase flow simulation. ASME 2009 fluids engineering division summer meeting, American Society of Mechanical Engineers, Vail, Colorado, USA, 515–523.
Roudnev, A. S., & Kosmicki, R. J. (2007). Effects of CFD modeling configuration on centrifugal slurry pump casing wear prediction. The 17th international conference on the hydraulic transport of solids, 7–11 May 2007, Cape Town, Southern African, 271–280.
Saito, S. (1982). The transient characteristics of a pump during start up. Bulletin of JSME, 25(201), 372–379.
Solutions DEM. (2010). EDEM 2.3 user guide. Edinburgh: DEM Solutions.
Tsuij, Y., Kawaguchi, T., & Tanaka, T. (1993). Discrete particle simulation of two-dimensional fluidized bed. Powder Technology, 77(1), 79–87.
Tsunamoto, H., & Ohashi, H. (1982). Transient characteristics of a centrifugal pump during starting period. ASME Journal of Fluid Engineering, 104(1), 6–13.
Wilson, K. C., Addie, G. R., Sellgren, A., & Clift, R. (Eds.) (1997). Slurry transport using centrifugal pumps. London: Blackie Academic & Professional.
Zhang, Y., Li, Y., Cui, B., Zhu, Z., & Dou, H. (2013). Numerical simulation and analysis of solid-liquid two-phase flow in centrifugal pump. Chinese Journal of Mechanical Engineering, 26(1), 53–60.