Research Article　　　　　　　　　　　　　　　　　　　　　　　　　　　　　**Open Access**

CFD Validation for Forces on Immersed Tubes in Fluidized Bed

Sushil Kumar*

Corporate R&D, BHEL, Hyderabad, Telangana, India

Abstract

Fluidized bed is widely used in chemical processing industries and heat transfer operations because of high heat transfer and mass transfer rates due to intermixing. This work deals with estimation of the force imparted by the bed material on the tube assemblies in a fluidized bed by CFD technique. The gas-solid flow was simulated by means of a multi-fluid Eulerian model incorporating the kinetic theory for solid particles. The pulsating forces were compared to experimental data of Kennedy et al. A good approximation was observed in force exerted over different tubes. However minor differences were observed in interval duration of the peak forces.

Keywords: Fluidized bed; CFD; Gas-solid flow; Pulsating forces

Introduction

Fluidized beds with immersed tubes are extensively used in industries. These tubes provide large surface area for heat transfer with the bed materials. Fluidized bed encounters time varying forces of an irregular nature due to the action of gas bubbles and turbulence within the bed. These forces contribute to vibrations which can lead to failures of the tube and their support system. To have a basis for structural design, a definition of load environment is required. Information on the magnitude and frequency composition of the applied forces is necessary in order to predict the fatigue life of a structure. The objective of this investigation is to validate the fluent capability of predicting the forces of the tubes. Forces predicted are compared with the experiment performed by Kenndy et al. [1]. In the experiment the forces were measured on various length of tube and ranges of fluidization conditions. Three beds considered had following dimension 0.3 × 0.3 m (1 ft by 1 ft), 0.91 × 0.91 m (3 ft × 3 ft) and 2.4 m × 0.3 m (8 ft × 1 ft). Out of these 3 bed, a bed with dimensions of 0.91 m × 0.91 m was chosen for our investigation as it has aspect ratio of industrial fluidized bed. Also for this case there were more experimental data available to compare with CFD results.

Experimental Setup

The fluidized bed test facility bed in experiment by Kennedy et al. [1] had various cross-sectional dimensions. This fluidized in the present discussion had a cross section of 0.91 m × 0.91 m (3 ft × 3 ft) and height of 1.4 m (4.5 ft). Here fluidizing air is supplied by a rotary, positive displacement blower with maximum capacity of 3.78 m³/sec at 51.7 kPa (8000 cfm at 7.5 psi). Air from blower enters a plenum below the air distributor which consists of perforated plate. Air flow is monitored with a venturimeter and superficial gas velocity is controlled by speed of the blower. Figure 1 shows the arrangement of tube array used in the experiment. Tubes are numbered in order 1 to 8 from the top where instruments were fixed in the experiment to capture the force data. Tubes of 5 cm OD were fixed in fluidized bed with fixed pitch as shown in Figure 1. In the experiment forces on the individual tubes were measured by supporting each end of a 5 cm (2 inch) diameter tube with strain gauze that were designed and built for this specific application. These load cells can measure both the vertical and horizontal components of force transmitted by tubes to its end support.

CFD Modeling

Traditionally, two approaches are followed for the study of multiphase flow phenomena in CFD: Eulerian–Lagrangian and the Eulerian–Eulerian. The first method is a fundamental approach which involves the balance of forces that act upon each of the particles and requires considerable computational effort. This approach also takes into account a collision model for commending the energy dissipation caused by the non-ideal particle-particle interactions. Thus this approach is limited to only fluidized bed of small in size and with very few particles. The Euler–Euler approach considers the dispersed phase (bed particles) as a continuous phase and is based on the Navier–Stokes equations applied to each phase. It is a more realistic approach for investigation of fluidized bed of Industrial scale [2,3]. An important aspect which influences the accuracy of CFD results for fluidized beds is the methodology used to extract the data. The data extraction methodology and the subsequent results, such as bed properties, bubble characteristics and bed expansion, suffers from great variation between studies reported in the literature [4]. Also, according to Asegehegn et al. [4] data extraction can have as much influence as the use of different constitutive relationships. For instance, Hulme et al. [5] shown that different volumetric solid fraction at the inlet led to different bubble average parameters (Table 1).

Modelling and meshing

A geometrical CAD model of 3 × 3 ft. fluidized bed with height of 2.8 m is imported in Ansys ICEM CFD. The height of the bed is doubled here in the model so that fluidized particles does not come out of the top section of the bed during start of the fluidization process in CFD. Tubes with diameter of 5 cm are laid as per experimental setup show in Figure 1. Meshing of the fluid zone was carried out with hexahedral elements. To capture the boundary layers around the tubes, fine prism mesh layers were created surrounding the tubes as shown in Figure 2. Total mesh elements for the fluid zone was approximated at 0.3 million. Mesh was imported in the Ansys fluent software for defining the problem setup. The bed was initially patched with 0.8 mm spherical particles with density of 2700 kg/m³ such that all the tubes are submerged. A uniform velocity of 1.5 m/s is applied at the bottom of the fluidized bed as inlet

*Corresponding author: Sushil Kumar, Corporate R&D, BHEL, Vikasnagar, Hyderabad-500 093, Telangana, India, Tel: +914023182359; E-mail: sushilkumar@bhelrnd.co.in
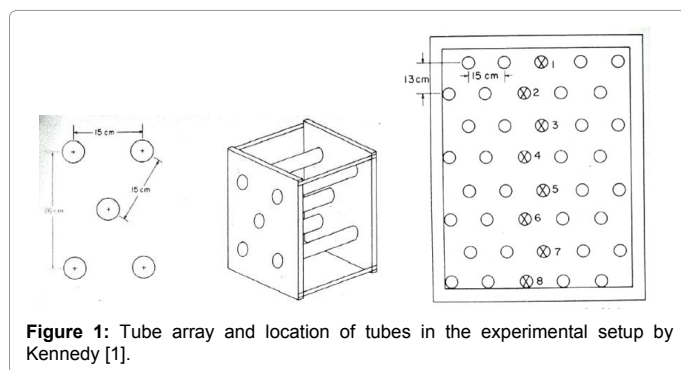
Figure 1: Tube array and location of tubes in the experimental setup by Kennedy [1].
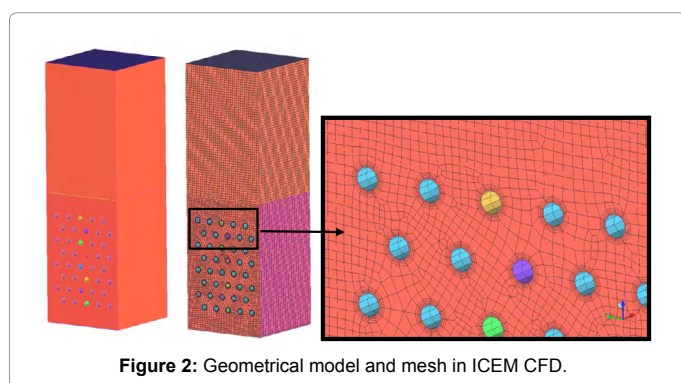
Figure 2: Geometrical model and mesh in ICEM CFD.

Parameter	Model (Ansys Fluent)	Reference
Granular viscosity	Syamlal-Obrien	[6]
Granular bulk viscosity	Lun et al.	[7]
Frictional viscosity	Schaeffer	[8]
Frictional pressure	Based-ktgf	[7]
Solid pressure	Lun et al.	[7]
Radial distn function	Lun et al.	[7]
Drag law	Syamlal-Obrien	[6]

Table 1: Closure equations.

boundary condition. The fluidized bed system was isothermal at 300K and initial pressure was set at 1 bar. The minimum void fraction of bed packing was set to 0.45. Top of the fluidized chamber was considered as pressure outlet boundary condition. The particle gas interaction was characterized by Syamlals- Obrien Drag law [6]. The coefficient for the drag law are obtained from the literature and Fluent theory guide based on particle size and phase. The particle- particle and particle-wall interaction was considered to be elastic in nature with coefficient of elasticity as 0.9. The simulation is initiated with time step of 1e-4 s and is simulated for 5 sec of real fluidization time. The QUICK and second order upwind were used for the spatial discretization of the continuity and momentum equations respectively while time was discretized using first order implicit. The Phase-Coupled SIMPLE algorithm was used for the pressure-velocity coupling.

Results and Discussion

This simulation was performed for 4 seconds of real flow time. The initial 1.5 sec were neglected to reduce the effect of divergence in some of the cell zones during the bed expansion. Once the bed is fully fluidized. The properties like forces, pressures and volume fraction of solids are produced in Ansys Fluent 15. In literature there is no distinct definition for bubble boundaries, but many investigators have

considered solid volume fraction equal to 0.2 [7-9] and same has been adopted here too. From the solid volume fraction contour plots as shown in Figure 3 it was observed that small bubble forms when air enters into the bed though distributor plates. These bubble grows in size while rising up in the bed and encounters tubes above it. There were two major observations regarding the bubble rise. First, the bubble is hitting the tubes encounter its path while rising and breaks into small bubbles; second it elongates vertically between the tubes and passes between the tubes without breaking. Sizes of some bubble increase also by coalescence with other bubbles. Finally, when bubble crosses all the rows of tubes it erupts at the top of the bed. The solid volume fraction contour below shows the same phenomenon happening inside the bed at any time t. Forces were monitored on each of the eight tubes marked in the Figure 1. Vertical forces for tube 6 and tube 8 were measured and is shown in Figures 4 and 5. The force time history here consists of pulses with durations of approximately of 0.2 s which closely matches with the pulses in experimental force data. In the Figure below it can be observed that forces on tube 6 and 8 are almost similar in nature. The force appears as pulses occurring at a rate of 2-3 per second with magnitude of around 250-350 N. In experimental data shown the peak forces vary from 200-350 N. Pulses on tube 8 (lower tube) precede those on the tube 6 (upper tube) by 0.1- 0.2 s. The distance between these tubes is 26.4 cm. Thus it can be concluded that pulse is propagating upwards with a velocity of 1.3-2.6 m/s. This velocity corresponds to bubble rise velocity predicted by CFD, while those by experiment is in the range of 1-1.7 m/s. The total vertical force predicted by CFD on the eight tubes is shown in Figures 6 and 7 below. The pulses were of magnitude 700 to 900 N with duration of around 0.8-1 sec. whereas in the experiment, forces were of magnitude 600 N to 800 N with a duration of 0.5-0.8 s. This gives a near match of CFD predicted results with the experimental data.

Conclusion

Hydrodynamics simulation were carried out to validate the experimentally determined forces on the tube to with the CFD simulated forces. The two fluid Eularian-Eularian model available in Ansys Fluent were able to simulate the bubble behavior in the fluidized bed. Tubes immersed in the bed significantly affect the bubble shapes and sizes which eventually decreases the mean diameter of bubble and bubble rise velocity. Forces on the tubes predicted through CFD matches trend of the experimental determined forces. Although there is reverse trend in the simulated forces pattern, as tubes at bottom of bed is exerting slightly lesser force than those above it. A complete information of experimental test facility bed is not available, which could have given a more accurate modelling of the test bed in CFD. However, the forces predicted in CFD occurs in pulses of 2-3 per second with magnitude of 250-350 N matches nearly with the experiment. This complex mechanism of bubble movement around the tubes, which alters the bubble aspect ratio can be the main reason for deviation of force predicted by the simulation and experiment. Deviation could also be due to the approximation of uniform velocity at the bottom of bed as nozzle details of distributor plate were not available. An intensive investigation of fluidized bed is needed to verify the mechanism of bubble rise in presence of tubes and forces acting upon the tubes. Further, a close experimental and numerical studies with dense tube arrangements can reveal more insight about the bed. Moreover, CFD modelling was capable of predicting the forces on the tubes and thus provides a basis in structural design of the fluidized bed tube bundles.
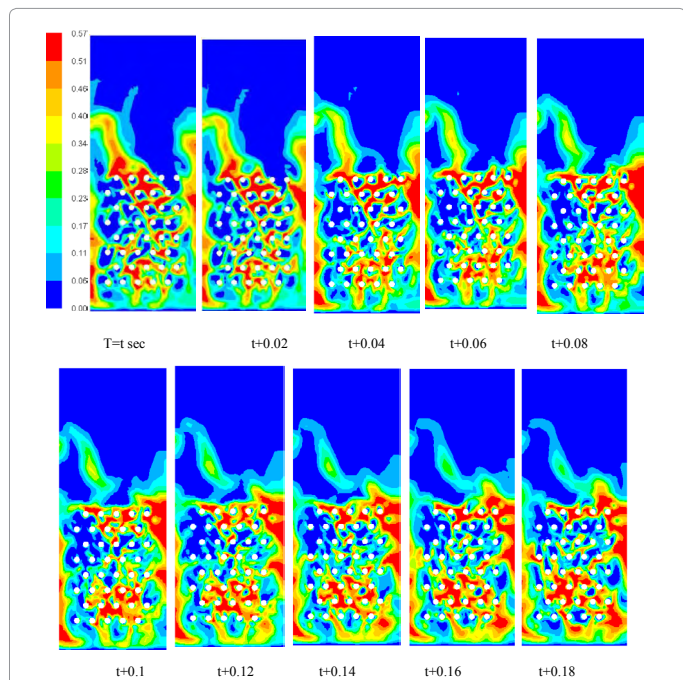
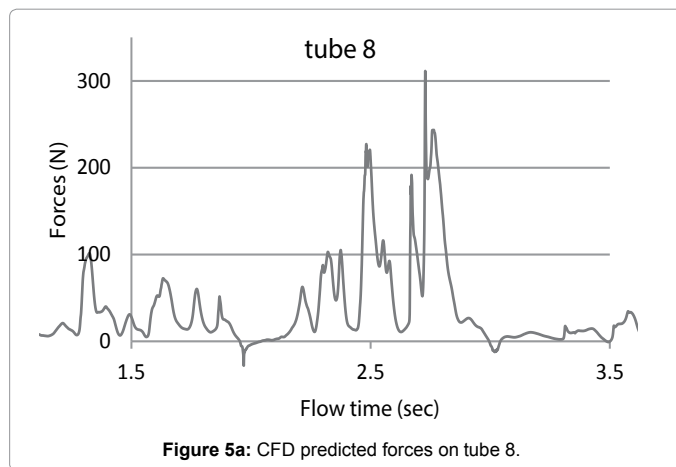**Figure 3:** Volume fraction contours for solids in a mid-plane of fluidized bed.

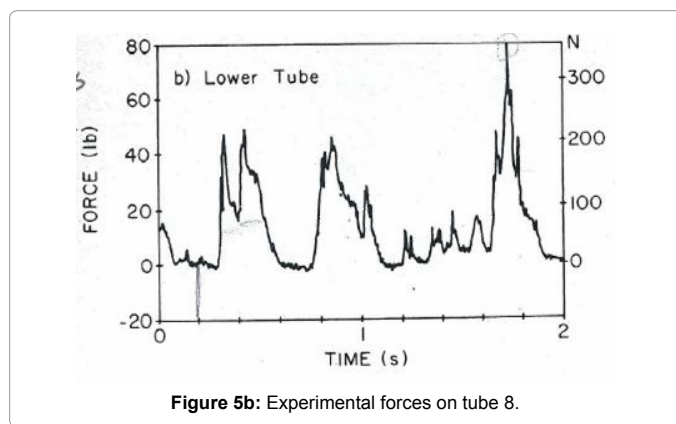

Figure 5a: CFD predicted forces on tube 8.
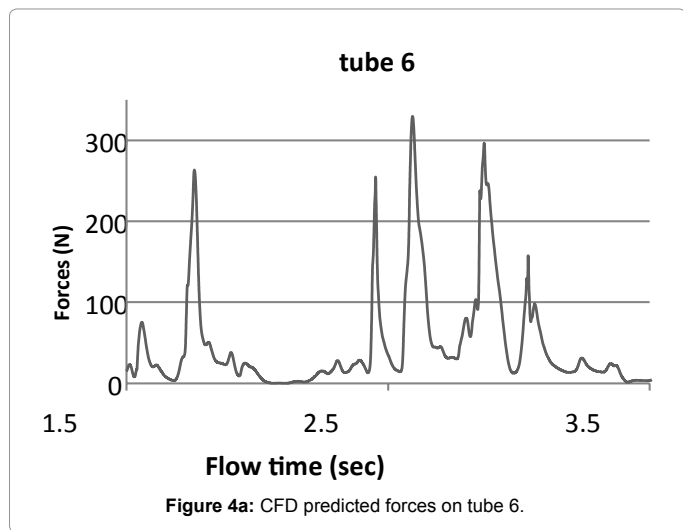

Figure 5b: Experimental forces on tube 8.

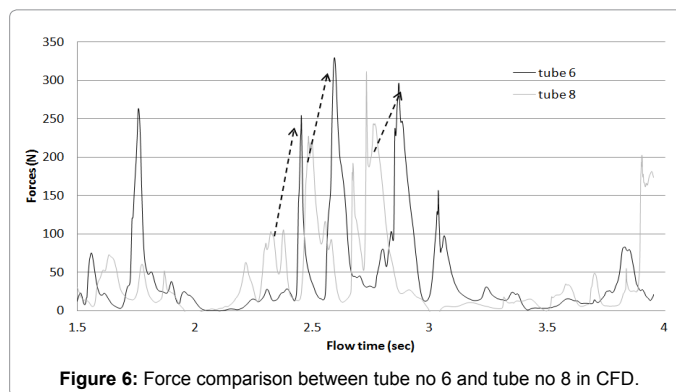**Figure 4a:** CFD predicted forces on tube 6.

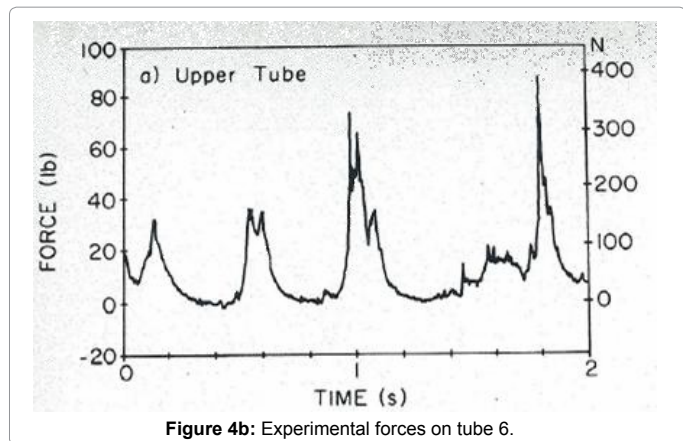**Figure 6:** Force comparison between tube no 6 and tube no 8 in CFD.

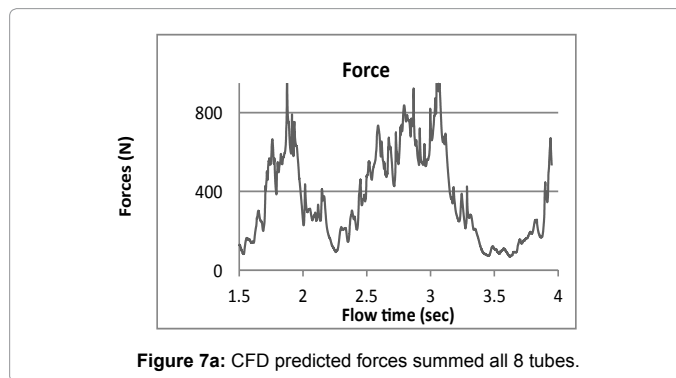**Figure 4b:** Experimental forces on tube 6.

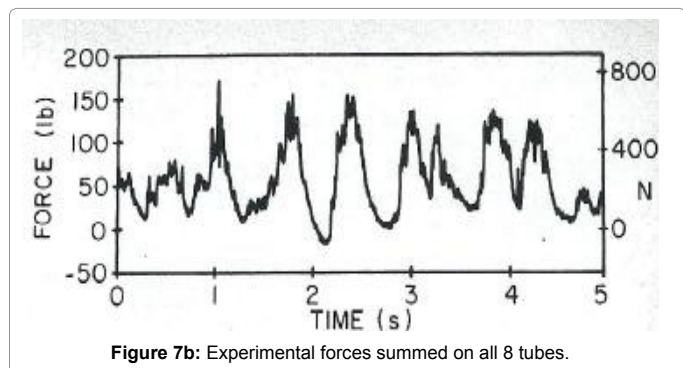**Figure 7a:** CFD predicted forces summed all 8 tubes.

Figure 7b: Experimental forces summed on all 8 tubes.

References

1. Kennedy TC, Donovan JE, Trigas A (1981) Forces on immersed tubes in Fluidized bed. AIChE Journal 27: 31-357.

2. Van Wachem BG, Schouten JC, Van den Bleek CM, Krishna R, Sinclair JL (2001) Comparative analysis of CFD models of dense gas–solid systems. AIChE Journal 47: 1035-1051.

3. Van der Hoef MA, van Sint Annaland M, Deen NG, Kuipers JA (2008) Numerical simulation of dense gas-solid fluidized beds: a multiscale modeling strategy. Annu Rev Fluid Mech 40: 47-70.

4. Asegehegn TW, Schreiber M, Krautz HJ (2011) Numerical Study of a Gás-Sólid Fluidized Beds Hydrodynamics: Influence of Immersed Horizontal Tubes and Data Analysis. International Journal of Chemical Reactor Engineering 9: 1-39.

5. Hulme I, Clavelle E, van der Lee L, Kantzas A (2005) CFD Modeling and Validation of Bubble properties for a Bubbling Fluidized Bed. Industrial e Engineering Chemistry Research 44: 4254 4266.

6. Syamlal M, O'Brien TJ (1989) Computer simulation of bubbles in a fluidized bed. AIChE Symposium Series 85: 22-31.

7. Lun CK, Savage SB, Jeffrey DJ, Chepurniy N (1984) Kinetic Theories for Granular Flow: Inelastic Particles in Couette Flow and Slightly Inelastic Particles in a General Flow Field. Fluid Mech J 140: 223-256.

8. Schaeffer DG (1987) Instability in the Evolution Equations Describing Incompressible Granular Flow. Journal of Differential Equations 66: 19-50.

9. Hulme I, Clavelle E, van der Lee L, Kantzas A (2005) CFD modeling and validation of bubble properties for a bubbling fluidized bed. Ind Eng Chem Res 44: 4254-4266.