Powder Technology 273 (2015) 19–32

Contents lists available at ScienceDirect

Powder Technology

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

Performance of vertical diffusers carrying Gas-solid flow: experimental and numerical studies

W.A. El-Askary [a,*], K.A. Ibrahim [a], S.M. El-Behery [a], Mofreh H. Hamed [b], M.S. Al-Agha [b]

[a] Mechanical Power Engineering Department, Faculty of Engineering, Menoufiya University, Shebin El-Kom, Egypt
[b] Faculty of Engineering, Kafrelsheikh University, Kafrelshiekh, Egypt

ARTICLE INFO

Article history:
Received 5 June 2014
Received in revised form 10 December 2014
Accepted 13 December 2014
Available online 19 December 2014

Keywords:
Gas-solid flow
Diffuser performance
Eulerian-Lagrangian
Four-way coupling
Chen-Kim k-ε turbulence model

ABSTRACT

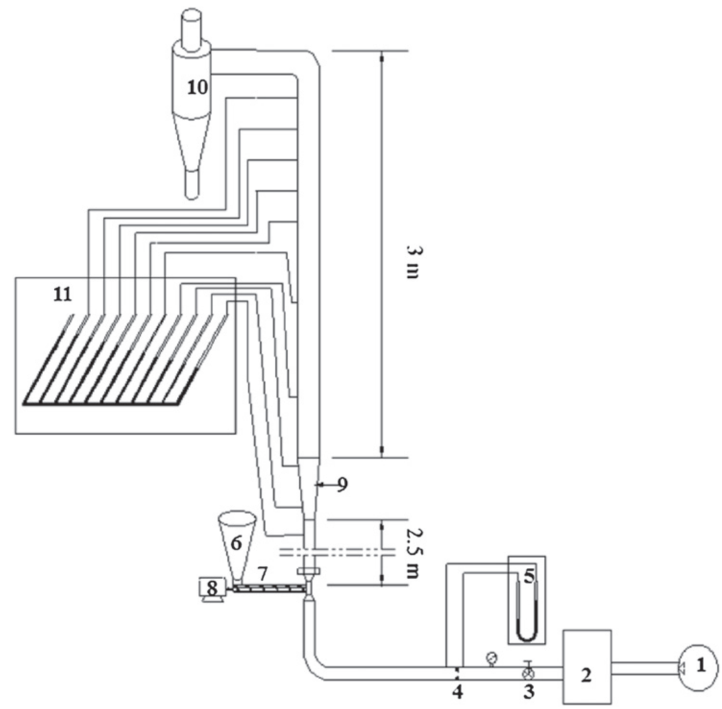
In the present work, numerical and experimental studies are carried out to investigate the performance of vertical straight-walled conical diffusers carrying air-solid two-phase flow. Eulerian-Lagrangian approach is used to numerically simulate the two phases using the Chen-Kim k-ε turbulence model. The continuous phase (gas) is simulated using Eulerian frame by solving Reynolds-Averaged Navier-Stokes equations (RANS), while the dispersed phase (solids) is simulated using particle tracking method. Coupling between the two phases is established by adding particle source terms and void fraction in the continuous phase equations. A 4-way coupling is adapted to include the effect of particle-particle collisions. Lift forces, particle dispersion and particle-wall collisions are also considered in the simulation of solid-phase. The experimental study is carried out on a pilot scale vertical pneumatic transport system. Four different diffuser geometries are tested at various inlet-Reynolds numbers and mass loading ratios. Sand particles of different sizes and mass flow rates are used to represent the solid phase. Comparisons between numerical predictions and experimental results indicated good agreement. The effects of solid parameters are significant for small angled-diffusers and decrease as the diffuser angle increases. It is also found that, there is a significant decrease in the separation zone within diffuser due to the presence of solid particles. Energy is transferred from the gas phase to the solid phase in the upstream pipe. This energy is transferred again to the gas phase through the diffuser and its downstream tangent pipe. The rate of energy transfer is enhanced by increasing the solid mass flow rate and decreasing the particle size. The results show that the mass loading and size of solid particles have significant effects on the diffuser loss coefficient.

1. Introduction

Diffusers are used in many fluid machines such as turbo machines, aircraft intakes, combustors, etc. The performance of these machines depends greatly on the diffuser performance. So there is a need to understand the behavior of such diffusers under various circumstances. Diffusion phenomenon in single phase flow has been studied by many researchers. The structure of turbulence in Azad diffuser (axisymmetric, conical, total divergence angle of 8°, an area ratio of 4:1 and a fully developed pipe flow at the inlet) is experimentally studied by Okwuobi and Azad [1]. Direct Numerical Simulations (DNS) were carried out by Lee et al. [2] to investigate the turbulence statistics and coherent structures in a conical diffuser. Fessler and Eaton [3] measured the particle velocity and concentration statistics in a vertical downward planar sudden expansion flow. The study focused on large-eddy particle Stokes numbers ranged from 0.5 to 7.4. The response of dense particles to the turbulent flow field after a sudden expansion has been investigated. Gas-phase velocities were measured in the presence of three different particle classes (90 and 150 μm diameter glass spheres and 70 μm diameter copper spheres). Yu et al. [4] used Large Eddy Simulations (LES) to investigate particle-laden turbulent flow over a backward-facing step. The study accounted only for the effect of drag and gravitational forces on particle motion. Good agreement was found between obtained predictions and the experimental data reported in [3]. Kubik and Kleiser [5] investigated the turbulence modification by particles in particle-laden flow in a backward facing step. The study was carried out at moderate Reynolds numbers using the DNS. A particle tracking method was used to predict the particles' motion. Effects of drag, gravity and lift forces were considered in the model. It has been reported that the degree of fluid phase turbulence modification depends on the particle Reynolds number as well as the Stokes number. De Souza et al. [6] carried out a numerical investigation of gas–particle flow in a vertical diffuser using four-way coupling Eulerian-Lagrangian approach. The separated flow region in the vicinity of the diffuser wall was found to be mitigated or eliminated, depending on the mass loading and the diffuser's wall characteristics.

* Corresponding author. Tel.: +20 2 0100 5255817, +20 2 048 3486965; fax: +20 2 048 2235695.
E-mail addresses: wageeh_elaskary@yahoo.com, Wageeh.Elaskary@sh-eng.menofia.edu.eg (W.A. El-Askary).

http://dx.doi.org/10.1016/j.powtec.2014.12.023
0032-5910/



1-Compressor
2-Air reservoir
3-Flow control valve
4-Flow meter (orifice plate)
5-U-tube mercury manometer
6-Solids reservoir
7-Screw feeder
8-Electrical motor
9-Tested diffuser
10-Cyclone separator
11-Multi-tube inclined manometer

Fig. 1. Schematic sketch of the experimental setup.

In the present study, a theoretical model that accounts for the effects of particle-particle and particle-wall collisions has been developed. A finite volume method and particle tracking approach is followed to develop a computer code. An experimental work is carried out on a pilot scale pneumatic conveying system. The effects of diffuser's angle, area ratio, inlet Reynolds number, solids mass loading ratio and particles size on the diffuser's performance are experimentally investigated. The flow separation and diffuser's performance under different inflow and geometrical conditions are also studied.

2. Experimental set-up

A schematic sketch of the experimental apparatus is illustrated in Fig. 1. The operation proceeds in the following sequence:

The air reservoirs (2) are fully charged using the compressor (1). The flow control valve (3) is adjusted to achieve the desired flow rate. The pressure and temperature upstream of the orifice meter (4) are recorded and the pressure drop across the orifice is measured using the U-tube mercury manometer (5). The electrical motor (8) is switched on to

Table 1
Diffusers dimensions used in the present study.

	α (degree)	D_1 (mm)	D_2 (mm)	L (mm)	Area ratio (AR)
Diffuser 1	3.4	54	104	420	3.71
Diffuser 2	7	38	104	268	7.11
Diffuser 3	7	54	104	200	3.71
Diffuser 4	7	63.5	104	155	2.56

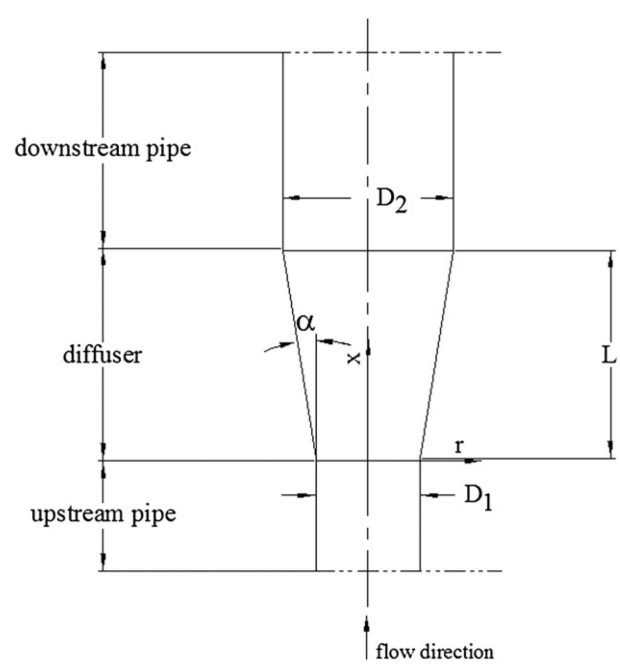


Fig. 2. Diffuser geometry.