Powder Technology 266 (2014) 22–37

Contents lists available at ScienceDirect

Powder Technology

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

Numerical analysis of particle flows within a double expansion

A.I.J. Love *, D. Giddings, H. Power

Division of Energy and Sustainability, The University of Nottingham, University Park, Nottingham, NG7 2RD, United Kingdom

A R T I C L E   I N F O

Article history:
Received 3 September 2013
Received in revised form 28 May 2014
Accepted 31 May 2014
Available online 11 June 2014

Keywords:
Computational fluid dynamics
Two-phase flow
Pulverised fuel
Coanda effect

A B S T R A C T

The effect of solid particles within flows having zones of recirculation is of interest in pulverised fuel distribution and combustion at burners. Previous modelling of a 1/4 scale test rig was performed by Giddings et al. (2004), and an instability was later identified within the domain. Subsequently the transient dynamics of the flow of air through a double expansion were investigated numerically and a recirculation zone was found to develop at one of the four corners of the expansion. In the work presented here the flow of solid particles through this double expansion is investigated using the commercial software ANSYS FLUENT R14.0. The Stress-Omega Reynolds Stress Model is used to model the gas phase turbulence and the Discrete Particle Model is used to model the solid particle flow. The dynamics of the flow are reported here for 10 μm and 60 μm particles and for mass loadings from 0 to 1 $kg_{particles}/kg_{air}$. The simulations show a distinct transition to a vortex shedding type instability with the addition of the discrete phase. Furthermore, for increasing mass loading and particle Stokes number the Coanda effect is reduced leading to two large recirculation zones in opposing corners of the domain. The characteristics of the flow field are in qualitative agreement with studies of particle flows in jet flows and shear layers. This work serves to highlight some of the challenges in modelling complex pneumatic conveying flows from an industrial perspective.

1. Introduction

The steady pneumatic transport of solid particles is of importance to many industries. This is particularly true of coal fired power plants where coal is first ground in the mills before being pneumatically transported to a number of burners through a convoluted system of pipes and splitters. In order to ensure good performance a steady homogeneous supply of fuel to the burners is desirable. The presence of flow features, such as a recirculation zone at expansions, further complicates the picture. For example, the study by Kuan and Yang [1] highlighted the unsteady flow downstream of a bifurcation. In addition, the interaction of pulverised fuel with a recirculation is key to the stability and NO_x performance of low NO_x burners. Computational fluid dynamics is utilised within industry to model such flows where simplifications, such as steady state analysis, are common but may not result in numerically stable solutions. In this study the flow through a double expansion is investigated and the effect of the discrete phase on the flow is evaluated.

There are a number of examples in the literature of flow through circular-to-rectangular transition ducts [2–4]. Patrick and McCormick [2] found asymmetric flow within a circular-to-rectangular transition duct. Miau et al. [3] went on to investigate the flow field in three transition ducts of constant cross sectional area, but different lengths, over a range of turbulent Reynolds numbers. A separation bubble, or stall region, was found in one corner for the lowest Reynolds number case, disappearing at higher Reynolds numbers. Characteristics of the flow depended primarily on the geometry. However, diffuser geometries do not maintain a constant cross sectional area but expand in two or three dimensions. Therefore, a greater adverse pressure gradient is present and boundary layer separation is more likely. Turbulent flow through two dimensional diffusers can generate a number of flow regimes: attached flow to both divergent walls, unsteady transitory stall, fully developed large stall/recirculation region, a hysteresis region and jet flow regime [5]. The transitory stall regime is characterised by the formation of a large recirculation in one corner.

The flows over backward facing steps, through diffusers and around bluff bodies are subject to a number of flow phenomena including separation, the Coanda effect, jet precession and vortex shedding. The Coanda effect is the deflection of a jet due to the generation of a low pressure region next to the jet as it entrains fluid [6], and precession being the rotation of a jet around an axis [7]. Reynolds Averaged Navier Stokes (RANS) turbulence models have been shown to capture the main flow features in each case. For example, the separation bubble in 2D and 3D diffusers has been captured by Reynolds Stress Models (RSMs) [8–10], Guo et al. [7] reproduced jet precession in an axisymmetric expansion, whilst Lardeau and Leschziner [11] and Iaccarino et al. [12] captured vortex shedding behind bluff bodies. Large Eddy Simulation (LES) models the smallest turbulent scales using sub-grid scale models

* Corresponding author.
E-mail addresses: enxail@nottingham.ac.uk (A.I.J. Love), donald.giddings@nottingham.ac.uk (D. Giddings), henry.power@nottingham.ac.uk (H. Power).

http://dx.doi.org/10.1016/j.powtec.2014.05.057

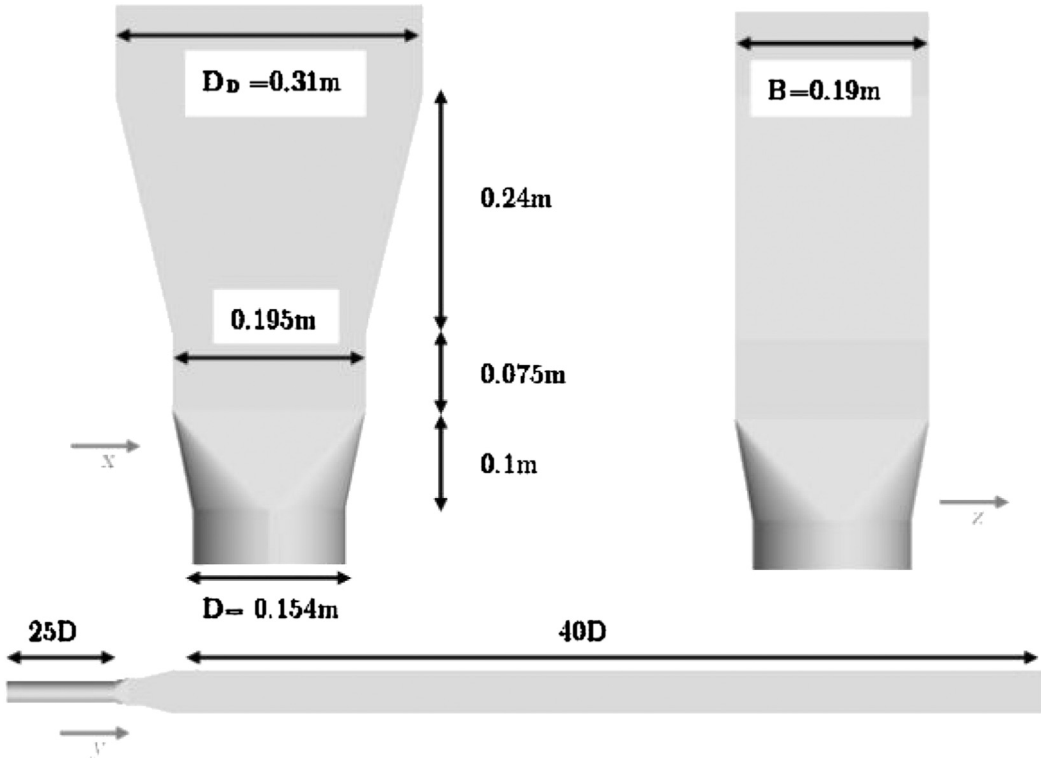

Fig. 1. Dimensions of the diffuser geometry.

and is capable of capturing both the random fluctuations in the fluid velocity and the largest turbulence structures. This approach has been utilised with good results for diffuser and bluff body flows [13,14]. Despite this LES still represents considerable computational expense in comparison to RANS models for industrially relevant confined flows at high Reynolds number [15]. By formulation RANS models average out the random velocity fluctuations, modelling the transport of turbulent kinetic energy and its dissipation, but have been found to capture the largest, deterministic scales of turbulence such as vortex shedding [12,11]. Furthermore, Garnier et al. [10] showed that unsteady RANS was capable of reproducing the reduction in the re-attachment length for a separated flow by periodic forcing at the optimum frequency. Although URANS underpredicted the reduction in recirculation length compared to LES especially at high forcing frequency as URANS filters the smallest turbulent fluctuations, from a practical point of view the prediction of the separation bubble and forcing response was found to be adequate.

The characteristics of particles and their impact on such complex flows have been investigated both experimentally and numerically. The dispersion of particles at a recirculation downstream of a backward facing step was investigated experimentally by Hardalupas [16]. Particles only enter the recirculation zone due to turbulent dispersion as they must cross the dividing streamline. They found bimodal probability distribution functions for both axial and inclined velocities near the shear layer for a large eddy Stokes number of 1. Particle concentration in the recirculation zone increased abruptly for a Stokes number of 1. Eaton and Fessler [17] provide an overview of preferential particle concentration by turbulence. It has been shown for many flows that small particles concentrate on the outside and between coherent vortex structures. Horender and Hardalupas [18] used PIV to gather measurements in a horizontal plane shear layer laden with mono-dispersed glass particles of sizes 55 and 90 μm. The experimental results showed that large eddies centrifuged particles into the low speed side of the flow, contributing to large particle velocity fluctuations. In addition, bimodal

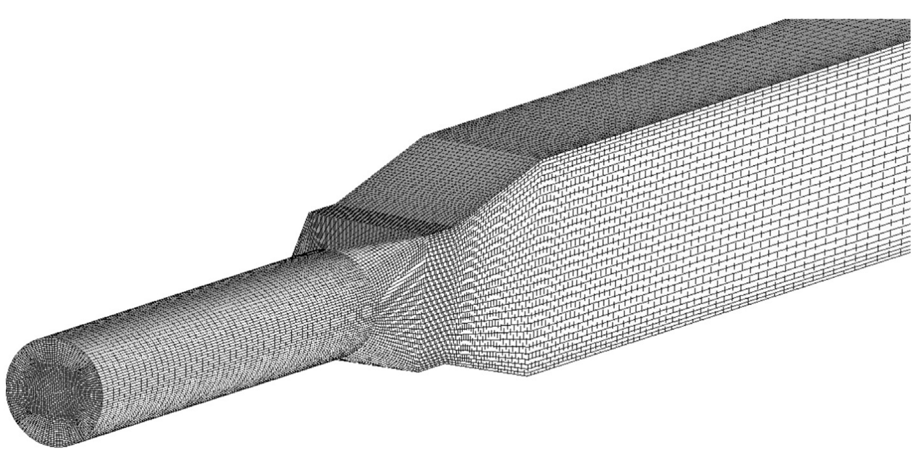

Fig. 2. Example of computational grid.