International Journal of Multiphase Flow 45 (2012) 70–90

Contents lists available at SciVerse ScienceDirect

International Journal of Multiphase Flow

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

One-way, two-way and four-way coupled LES predictions of a particle-laden turbulent flow at high mass loading downstream of a confined bluff body

M. Alletto, M. Breuer *

Professur für Strömungsmechanik, Helmut-Schmidt-Universität Hamburg, D-22043 Hamburg, Germany

ARTICLE INFO

Article history:
Received 8 February 2012
Received in revised form 16 May 2012
Accepted 17 May 2012
Available online 2 June 2012

Keywords:
Dispersed two-phase flow
Eulerian–Lagrangian LES approach
Two-way and four-way coupling
Deterministic collision model
High mass loading
Collision frequency

ABSTRACT

In the present contribution an eddy-resolving scheme (large-eddy simulation) is combined with an efficient particle tracking algorithm for individual particles and a deterministic collision model. The purpose is to set-up a reliable methodology for the prediction of complex particle-laden two-phase flows at high mass loadings. The objectives are two-fold. On the one hand the suitability of the entire method to tackle practically relevant turbulent flows should be proven. On the other hand the influence of the fluid-particle interaction (two-way coupling) as well as particle–particle collisions (four-way coupling) is investigated in detail. For both purposes this numerical study is aligned to the experimental investigation of the bluff-body configuration by Borée et al. [J. Borée, T. Ishima, I. Flour, The effect of mass loading and inter-particle collisions on the development of the polydisperse two-phase flow downstream of a confined bluff body, J. Fluid Mech. 443 (2001) 129–165]. In this set-up a fully developed pipe flow laden with polydisperse glass beads enters a cylindrical chamber with an outer annular confined flow without swirl. In contrast to previous numerical studies both mass loadings ($\eta = 22\%$ and 110%) and thus also inter-particle collisions are taken into account. Contrary to the experimental investigation the predictions allow to artificially isolate different physical effects in order to clarify their importance. Especially for the high mass loading case interesting new results about the role of fluid-particle interactions and particle-particle collisions are enlightened.

1. Introduction

Confined bluff-body flows are typical flow configurations relevant for a variety of practical applications involving particle-laden or droplet-laden turbulent flows especially for combustion devices. For example, in pulverized coal combustion primary air and coal particles are injected in the center of the combustion chamber and a secondary stream of air is introduced on the periphery. For this kind of configuration a cone-shaped body is located close to the exit of the primary air in order to generate a recirculation region to stabilizes the flame generated by the combustion of the coal particles (Xu et al., 1995; Shi et al., 1997). For non-premixed combustion the same stabilization measure is adopted to prevent that the flame is lifted-off or even extinguished: The fuel jet is injected inside a recirculation region generated by either a bluff-body flow (Schefer et al., 1987, 1994; Chen et al., 1990, 1998b) or a diverging flow with swirl (Chen et al., 1990).

A variety of experimental and numerical investigations on this topic are available in the literature. A few related to particle-laden two-phase flows should be mentioned here. Sommerfeld and Qiu (1991) studied the particle dispersion characteristics in a confined swirling flow with a swirl number of about 0.5 by applying the phase-Doppler anemometer technique. They measured both the particle and the fluid velocity as well as the particle sizes. In Sommerfeld and Qiu (1993) this experimental study was further extended and additionally complemented by numerical predictions based on an Euler–Lagrange approach using a Reynolds-averaged k-ε turbulence model and a one-way coupling for the low mass loading considered (see, also, Sommerfeld et al., 1992). Apte et al. (2003b) picked up the turbulent, lightly loaded particle-laden flow (case 1: $\eta = 3.4\%$) in a coaxial-jet configuration of Sommerfeld and Qiu (1991, 1993) and used these experiments to validate their unstructured two-phase flow solver based on LES. An efficient particle tracking scheme on unstructured meshes was developed to compute the disperse phase. A two-way coupling was applied considering the interaction between the continuous and particulate phase only by momentum exchange terms. Collisions among particles were assumed to be negligible. Good agreement with experimental data was obtained for both phases. Furthermore, the LES results (Apte et al., 2003b) were found to be significantly more accurate than the RANS predictions of the same problem (Sommerfeld et al., 1992; Sommerfeld and Qiu, 1993). Apte et al. (2003b) hold two reasons responsible for the success: (a) the effectiveness of LES in predicting turbulent mixing over RANS and (b) tracking of

* Corresponding author.
E-mail address: breuer@hsu-hh.de (M. Breuer).

0301-9322/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijmultiphaseflow.2012.05.005

large number of particles to obtain a dynamic representation of the disperse phase as opposed to the parcels approach generally employed in RANS-based simulations. More recently Oefelein et al. (2007) carried out LES predictions for the same case taking both mass loadings experimentally investigated by Sommerfeld and Qiu (1991, 1993) (case 1: $\eta = 3.4\%$ and case 2: $\eta = 17\%$) into account. Also for the higher mass loading inter-particle collisions were still neglected. Nevertheless, a good agreement for the mean and r.m.s values of the gas phase was observed. The particulate phase showed slightly larger deviations to the measurements but still within the experimental uncertainties.

Thus the general concept of applying an eddy-resolving scheme (LES) in combination with the tracking of many individual particles as used in the present study seems to be an adequate trend to follow. Furthermore, all these investigations using particles and not droplets appearing for liquid fuels are considered as an essential first step before the inclusion of effects such as atomization, evaporation and combustion.

The same applies to the following studies which contrary to the aforementioned investigations are assuming a combustion chamber without an azimuthal swirl velocity component at the inlet. In order to investigate the effect of mass loading and inter-particle collisions Borée et al. (2001) set up an experiment for a polydisperse two-phase flow in a model combustion chamber without swirl. A particle-laden central jet enters the cylindrical chamber surrounded by an annular confined ring flow. Using a two-component phase-Doppler anemometer particle size and velocity measurements of both phases were carried out. This experimental set-up defines an excellent test case for any kind of numerical method due to the following reasons:

- The geometry of the flow configuration is simple; nevertheless, the case involves multiple challenging flow features typical of combustion chamber flows: strong recirculating zones generated by the confined bluff-body geometry, multiple stagnation points and a high level of turbulent mixing.
- By assuming isothermal conditions (cold flow), no swirl at the inlet and solid spherical particles of high density instead of droplets (to avoid the modeling of coalescence, break-up, evaporation and combustion), the complexity of the set-up is reduced to a reasonable level. That allows to restrict the investigations to the dynamics of both phases and their interaction.
- Detailed experimental data are available for two mass loadings, a moderate value of $\eta = 22\%$ and a high level of $\eta = 110\%$. Consequently, different modeling assumptions (e.g., one-way, two-way and four-way coupling) can be applied and certified. At the moderate mass loading a two-way coupled simulation is generally thought to be sufficient, whereas for $\eta = 110\%$ most likely the inter-particle collisions can no longer be completely ignored.
- Owing to the fully developed pipe and annular ring flow entering the chamber, the inflow conditions are clearly defined which is a very important issue for this kind of flows and in general for a precise definition of a test case.
- Overall the case is highly attractive due to its combined relevance for practical combustion devices and fundamental understanding of turbulence and mixing (Borée et al., 2001).

As a consequence of these beneficial features the flow configuration has already been used as a benchmark case in a variety of investigations. Making no claim to be complete, in the following a brief summary of previous investigations of this case is provided.

Minier et al. (2004) carried out one-way and two-way coupled Euler–Lagrange computations, where the continuous phase is solved by means of the Reynolds-averaged Navier–Stokes equations combined with a Reynolds stress model. The motion of the particles is described by a particle PDF equation which is solved by a Monte Carlo method using a trajectory point of view. The PDF model is formulated as a particle stochastic Lagrangian model, i.e., a large number of stochastic particles are tracked which mimic the behavior of real particles dispersed in the fluid. The stochastic differential equation written in continuous time denotes a Langevin equation for the fluid velocity seen by the particles. Inter-particle collisions are not taken into account. Fluid and particle results for a mass loading of $\eta = 22\%$ tracking polydispersed particles with a mass-averaged diameter of $\overline{d_{p,M}} \approx 60$ µm are compared with the experiments of Borée et al. (2001). Good agreement was found between the simulation and the measurements. Considering a two-way coupled flow the locations of the stagnation points were shifted considerable downstream. Minier et al. (2004) pointed out that this indicates the necessity of further studies on the coupling between the particles and the fluid.

Fede and Simonin (2010) carried out 2D-RANS simulations based on an Euler–Euler approach. They adopted either a k-ε or a Reynolds stress model for the fluid phase. For the particulate phase either a two-equation or a second-order approach was used. A mass loading of $\eta = 22\%$ was considered without taking inter-particle collisions between classes into account. The Reynolds stress model gives results in good accordance with the experimental data of Borée et al. (2001). Furthermore, the comparison between a single particle class with an equivalent diameter and a polydisperse particle phase evidenced that the particle diameter has strong influence on the particle behavior.

Applying an unsteady Eulerian–Lagrangian approach with two-way coupling Chrigui et al. (2010) demonstrated the general feasibility of URANS based predictions for this case. A non-linear k-ε model and a Markov-sequence model for the dispersion of the particles and the interaction with the turbulent flow was used. The particles were assumed to be monodisperse choosing the mass-averaged diameter $d_p = \overline{d_{p,M}} \approx 60$ µm as the representative diameter for all present particle classes. In agreement with the other studies mentioned before, inter-particle collisions were not taken into account, since solely the moderate mass loading case was considered.

Riber et al. (2009) compared the results of three different LES solvers with the experimental results of Borée et al. (2001): an Euler–Euler explicit (EEE) compressible solver, an Euler–Lagrange explicit (ELE) compressible solver and an Euler–Lagrange implicit (ELI) incompressible solver. For the EEE and ELE solvers the inflow conditions were achieved by applying a mean turbulent profile superimposed by artificial turbulent fluctuations at a plane located $z/R_{\text{pipe}} = -10$ upstream of the chamber inlet. For the ELI solver the measured mean velocity profile for the annular flow without any fluctuations was imposed at the inlet of the computational domain. For the inner pipe a snapshot of a separate turbulent pipe flow was taken. Using Taylor's hypothesis the streamwise spatial abscissa was divided by the mean velocity in the pipe to transform it into the temporal abscissa. The resulting solution was then interpolated in space and time to obtain the inflow data. Only the case with moderate mass loading $\eta = 22\%$ was studied based on two-way coupling feeding the chamber with monodisperse 60 µm particles as the representative class. The initial mean particle velocity profile was set equal to the measured one. The releasing plane was equal to $z/R_{\text{pipe}} = -10, -9.5$, and -0.3 for the EEE, ELE and ELI solver, respectively. Riber et al. (2009) achieved good agreement with the experiments. Some deviations between the results of the different simulation methodologies were found and predominantly attributed by the authors to the different boundary conditions at the inlet, especially at the particle-laden pipe inflow.

In the present study LES predictions for an incompressible fluid combined with Lagrangian particle tracking of individual particles are carried out for this benchmark case. Compared to the paper of