Computers & Fluids 66 (2012) 52–62

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

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

Measurement and simulation of a droplet population in a turbulent flow field

Róbert Bordás [a,1], Volker John [b,c], Ellen Schmeyer [b,2], Dominique Thévenin [a,*]

[a] Lab. of Fluid Dynamics & Technical Flows, University of Magdeburg "Otto von Guericke", Magdeburg, Germany
[b] Weierstrass Institute for Applied Analysis and Stochastics, Leibniz Institute in Forschungsverbund Berlin e.V., Berlin, Germany
[c] Department of Mathematics and Computer Science, Free University of Berlin, Berlin, Germany

ARTICLE INFO

Article history:
Received 14 March 2011
Received in revised form 24 April 2012
Accepted 7 May 2012
Available online 30 May 2012

Keywords:
Two-phase turbulent flow
Disperse droplet population
Non-intrusive measurements
Population balance systems
Variational multiscale method

ABSTRACT

The interaction of a disperse droplet population (spray) in a turbulent flow field has been investigated by combining wind tunnel experiments with simulations based on a population balance system. The behavior of the droplets is modeled numerically by a population balance equation. Velocities of the air and of the droplets are determined by non-intrusive measurements. A direct discretization of the 4D equation for the droplet size distribution is used in the simulations. Important components of the numerical algorithm are a variational multiscale method for turbulence modeling, stabilized finite difference schemes for the 4D equation and a pre-processing approach to evaluate the collision integrals. The simulations of this system accurately predict the modifications of the droplet size distribution from the inlet to the outlet of the measurement section. Since the employed configuration is simple and considering that all measurement data are freely available thanks to an internet-based repository, the considered experiment is proposed as a benchmark problem for the simulation of disperse two-phase turbulent flows.

1. Introduction

In this paper detailed numerical simulations of systematic experimental studies concerning droplet populations interacting with a turbulent flow are presented. Such investigations are important to characterize modifications in the Droplet Size Distribution (DSD) resulting from droplet/droplet interactions induced by turbulent structures.

Water droplets with an initial diameter up to 50 μm are injected into a Göttingen-type wind tunnel with a closed test section. Velocities of both phases (air and droplets) are carefully determined by means of non-intrusive measurement techniques. In this way, suitable time-averaged boundary conditions and data are available to validate corresponding numerical simulations.

All measurement data are collected in an online database accessible at http://www.ovgu.de/isut/lss/metstroem. The raw measurement results are further post-processed, so that all required data are in a suitable form for comparisons and validation.

The behavior of the droplet population is modeled by means of a population balance system, consisting of the Navier–Stokes equations describing the air flow together with an additional equation for the DSD. In this last equation, the transport, growth, and coalescence of droplets is taken into account. The DSD describes the spatial evolution of the diameter of the droplets, the so-called internal coordinate, such that the equation for the DSD is finally defined in a 4D domain.

Population balance systems can be applied for modeling many processes in engineering and nature, like precipitation and crystallization processes, or rain formation. The development of accurate and efficient numerical methods for such simulations is an active field of research. Several suitable approaches have been proposed in the literature. In particular, moment-based methods like the quadrature method of moments (QMOM) [34], in which the equation in the 4D domain is replaced by a system of equations for the moments defined in the 3D flow domain, appears promising. A possible extension of QMOM is the direct quadrature method of moments (DQMOM) [33]. Further, operator splitting techniques have been studied recently [8], projecting the solution of the 4D problem onto the solution of a 1D problem followed by the solution of a 3D problem. Moment-based approaches and operator splitting schemes are beneficial, since the solution of the 4D equation is not needed any more. On the other hand, additional errors are introduced.

In the present paper, a direct discretization of the 4D equation for the DSD is retained, since the accuracy of the results is here more important than the numerical efficiency of the simulations.

* Corresponding author.
E-mail addresses: bordas@ovgu.de (R. Bordás), volker.john@wias-berlin.de (V. John), ellen.schmeyer@wias-berlin.de (E. Schmeyer), thevenin@ovgu.de (D. Thévenin).

[1] The work of R. Bordás was supported by Grant Th881/13-2 within the DFG priority programme 1276 MetStröm: Multiple Scales in Fluid Mechanics and Meteorology.
[2] The work of E. Schmeyer was supported by Grant Jo329/8-2 within the DFG priority programme 1276 MetStröm: Multiple Scales in Fluid Mechanics and Meteorology.

0045-7930/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2012.05.003

Simulations based on the 4D equation can be found rather rarely in the literature. However, with increasing hardware capabilities and with modern numerical methods, this is an attractive approach since it does not require any additional assumptions, e.g., for closing the system, and it does not introduce an additional modeling error. In the used method, the turbulent flow field is simulated fully implicitly. A variational multiscale (VMS) method is applied for turbulence modeling. Several stabilized finite difference methods combined with explicit time stepping schemes are used as temporal discretization for the population balance equation. A pre-processing approach was applied to compute the collision integrals. With this numerical approach, it will be shown that the experimentally observed evolution of the DSD between the inlet of the flow domain and its outlet can be reproduced accurately. The sensitivity of the prediction with respect to varying numerical methods is studied. To our best knowledge, the combination of the used methods for simulating a population balance system cannot be found so far in the literature.

The considered configuration corresponds to an experiment proposed as a benchmark problem for the simulation of population balance systems, since:

- all data are freely available in the online database at http://www.ovgu.de/isut/lss/metstroem,
- the considered geometry and setup is simple,
- first numerical studies are already available, supporting the accuracy of the experimental measurements.

The paper is organized as follows. The experimental setup is first described in Section 2, followed by the measurements and the post-processing procedure in Section 3. Section 4 describes the population balance system used to model the experiments. The numerical methods employed to simulate this setup are discussed in Section 5 and the simulation results are presented in Section 6. A summary is given at the end of the paper.

2. Experimental setup

A special wind tunnel available at the laboratory of Fluid Dynamics & Technical Flows has been used for the present experimental investigation of disperse two-phase flows corresponding to meteorological conditions found in cumulus clouds [4,5]. This wind tunnel can be used to investigate a variety of two-phase (air/liquid) flows [12], see Fig. 1. It is a fully computer-controlled, Göttingen-type wind tunnel. Operation with a closed test section enables the controlled and reproducible investigation of two-phase mixtures in the test section. The test section is of size $H \times W \times L = 500 \times 600 \times 1500$ mm. It includes a measurement section of cross-section 450×500 mm whose windows are optically transparent in the visible spectrum. In this manner non-intrusive measurements are possible, which is essential for high-quality experimental investigations of such flows.

The disperse phase was added to the air flow with the help of an injection system. The sprays were actuated by means of eccentric screw pumps. The number of revolutions per minute (rpm) was set with the help of a frequency regulator to a prescribed value by means of a Proportional Integral Derivative (PID) regulation coded in LabView®. In order to investigate rain formation and cloud droplet interactions a full cone pneumatic atomizing nozzle was used (Type 166.208.16.12 from the company Lechler GmbH), relying on the liquid pressure principle and applying an air gauge pressure of 1.2 bar [3].

Since the influence of the support of the injection system could be noticed especially in the upper half of the measurement section, the measurement area was finally restricted to the lower half of the cross-section, see Fig. 2. The resulting velocity inhomogeneity of the air flow was then below 5% with a turbulence intensity below 7% (mean value of 2.4%). The selected nozzle shows a typical six-hole spray pattern caused by the six orifices in the nozzle. In order to reduce the influence of this pattern, the water was injected in counter-flow direction. In this way, the droplets were more homogeneously distributed, and the relative velocity difference between continuous and dispersed flow at the entrance of the measurement section was decreased, suppressing to a large extent the six-hole pattern.

3. Measurement procedure

The longitudinal coordinate of the beginning of the measurement section is defined as $x = 0$ mm. Different measurement planes perpendicular to the main flow direction were investigated, at $x = 0$ mm, $x = 200$ mm, and $x = 400$ mm, see Fig. 1. The first plane at $x = 0$ mm was measured particularly thoroughly, since it provides the information needed for the boundary condition of the numerical simulations.

Experimental measurements were systematically carried out by means of non-intrusive measurement techniques. Therefore, a small quantity of suitable tracer particles must be added to the flow. Such particles follow the structures of the continuous phase much better [1,2] than the considered droplets, allowing an indirect measure of the gas flow properties. For this reason, the velocities of both phases were measured in two separate steps.

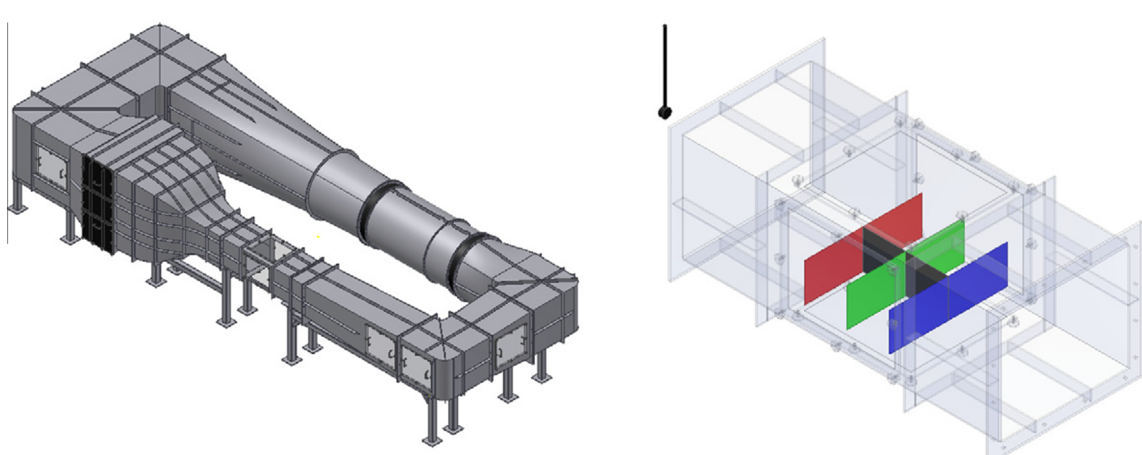

Fig. 1. Left: Göttingen-type two-phase wind tunnel with closed test section. Right: test section, with measurement planes colored.