Nuclear Engineering and Design xxx (2017) xxx–xxx

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Direct numerical simulation of a turbulent bubbly flow in a vertical channel: Towards an improved second-order reynolds stress model

Guillaume Bois

Den-Service de thermo-hydraulique et de mécanique des fluides (STMF), CEA, Université Paris-Saclay, F-91191 Gif-sur-Yvette, France

HIGHLIGHTS

- DNS of an upward turbulent bubbly flow in a plane channel is presented.
- Deformable bubbles are tracked using the Front-Tracking algorithm of TrioCFD.
- An up-scaling approach from DNS towards two-phase RANS CFD modelling is presented.
- Simulations of the averaged flow are performed within NEPTUNE_CFD.
- Turbulence models (SSG and EBRSM) are compared to the DNS reference data.

ARTICLE INFO

Article history:
Received 28 April 2016
Received in revised form 16 January 2017
Accepted 21 January 2017
Available online xxxx

Keywords:
Direct numerical simulation
Bubbly flow
Turbulence
Reynolds stress model

ABSTRACT

Two-phase turbulence has been studied using a Direct Numerical Simulation (DNS) of an upward turbulent bubbly flow in a so-called plane channel. Fully deformable monodispersed bubbles are tracked by a Front-Tracking algorithm implemented in TrioCFD code on the TRUST platform. Realistic fluid properties are used to represent saturated steam and water in pressurised water reactor (PWR) conditions. The large number of bubbles creates a void fraction of 10%. The Reynolds friction number is 180. Time- and space-averaging is used to compute the main variables of the averaged scale description (*e.g.* void fraction, liquid and vapour velocities...) along with the Reynolds stresses and the turbulent dissipation rate tensor. Altogether, they provide reference profiles to assess and further improve Reynolds Stress models. A low-Reynolds version of the SSG model (Speziale et al., 1991) called EBRSM (Manceau and Hanjalić, 2002; Manceau, 2005) is applied in the context of two-phase flows with additional interfacial production terms. The model has been implemented and tested in the two-fluid Euler-Euler model of NEPTUNE_CFD code. The comparison with DNS demonstrates that the interfacial momentum closure plays a dominant role over the turbulent closure hypothesis in the present physical conditions.

1. Introduction

Two-phase bubbly flows are found in many engineering applications. They involve a wide range of scales, from the Kolmogorov scale to the macroscopic flow structures, and in between, the bubble diameter. For industrial applications such as Nuclear Reactor Safety analysis, it is essential to correctly model the main characteristics of such flows. Originally, one-dimensional averaged models have been developed based on empirical correlations. Then, 3D-models that are averaged on finer space- and time-scales have been developed in the context of CMFD (Computational MultiFluid Dynamics) (Guelfi et al., 2007). In the quest of reduced uncertainties, the use of Reynolds Averaged Navier Stokes (RANS) two-fluid equations is the most reliable approach, but the accuracy of the predicted results depends on the constitutive relations used to close the turbulent and interfacial transfers. Those models rely on local correlations very difficult to establish based on experiments. Hence, in this paper, we focus on the realisation of an up-scaling approach from local-scale simulations towards two-phase RANS CFD modelling. This approach aims at extracting information (such as correlations) from fine-scale simulations in order to suggest or calibrate new models for the Reynolds stresses or the interfacial momentum transfer.

Direct Numerical Simulations (DNS) of two-phase flow being used as "numerical experiments" are then an excellent tool to develop local closures to the averaged models because they grant access to local quantities. In this paper, we present a first up-scaling step in which emphasis is laid on the turbulent fluxes, leaving aside the matter of interfacial transfers. The general

E-mail address: guillaume.bois@cea.fr

http://dx.doi.org/10.1016/j.nucengdes.2017.01.023
0029-5493/

Please cite this article in press as: Bois, G. Direct numerical simulation of a turbulent bubbly flow in a vertical channel: Towards an improved second-order reynolds stress model. Nucl. Eng. Des. (2017), http://dx.doi.org/10.1016/j.nucengdes.2017.01.023

Table 1

Review of turbulent bubbly flows simulations: main simulation parameters. N. A.: Not Available. Lengths are given either in wall unit defined from the Reynolds friction number ($1\text{w.u.} = h/\text{Re}_\tau$, also called *viscous unit*) or in dimensionless unit $1\text{d.u.} = h$ based on the channel half-width h. The inclination of the channel can be horizontal or vertical (upward or downward). When $g = 0$ has been reported, it indicates that the computation does not take gravity into account. n_b is the total number of bubbles tracked in the computational domain.

Author	Kanai and Miyata (2001)	Kawamura and Kodama (2002)	Lu et al. (2005)	Lu et al. (2006)	Lu and Tryggvason (2008)	Lee et al. (2014)	Bois G.
Domain	1	6.4	π	π	π	π or 2π	2π
size	1	2	2	2	2	2	2
(d.u.)	1	3.2	$\pi/2$	$\pi/2$	$\pi/2$	$\pi/2$ or π	π
Resolution	64	64	256	192	256	256 or 512	384
	64	64	256	160	192	192 or 192	1152
	64	64	128	96	128	128 or 256	192
Δy (w.u.) min/max	2.81	N. A.	0.21/4.67	0.79/2.19	0.21/4.67	0.21/4.67 or 0.42/9.3	0.3125
$\Delta x, \Delta z$ (w.u.)	2.81	N. A.	2.20	2.08	1.56	1.56	2.94
Fluids	Air/water	Air/water	Air/water	Air/water	Air/water	Air/water	Liq./vap.
Inclination	Horiz.	Horiz.	$g = 0$	Downward	Upward	Upward	Upward
Re_τ	180	180	180	127.3	127.2	127–280	180
D_b (w.u.)	28.8	71–113	54	31.8	38.3	38 or 19	36
D_b (d.u.)	0.16	0.4–0.63	0.4	0.25	0.30	0.3	0.2
D_b/Δ	10.2	4–12	11–326	14–40	8–231	8–231	12–115
n_b	27	9–54	16	18–72	21	21 or 84	942
α	6%	3–8.6%	5.4%	1.5–6.0%	3.0%	3.0%	10%

interest for industrial applications covers a wide range of very different flows, from classical single-phase turbulent flows, to very complex boiling flows (with many different topological regimes). As a first step away from single-phase turbulence, we focus on an adiabatic bubbly flow, between two infinite parallel walls. This article starts with a brief overview of existing DNS of turbulent bubbly flows (Section 2). Then, the characteristics of the test-case and the numerical method are described in Section 3. The two-fluid model is presented in Section 4 and applied to the DNS configuration in Section 5. Parametric studies on turbulence modelling are performed. Finally, conclusions and prospects are drawn in Section 6.

2. Overview: DNS of turbulent bubbly flows

The first DNS of single-phase flow between two parallel walls has been performed by Kim et al. (1987), for a friction Reynolds Number $\text{Re}_\tau = \rho u_\tau h/\mu$ of 180, where ρ is the density, h is the channel half-width, μ is the liquid viscosity and u_τ is the wall friction velocity defined by $u_\tau = \sqrt{\tau_w/\rho}$, where τ_w is the mean shear stress at the wall. Increasing computational power has enabled the rise of the Reynolds number to the value of $\text{Re}_\tau = 2003$ (Hoyas and Jimenez, 2006; Hoyas and Jimenez, 2008) and very recently $\text{Re}_\tau = 5200$ has been simulated (Lee et al., 2014).

DNS of turbulent two-phase flows are much more recent but it has already proven very useful to better understand the influence of non-dimensional parameters on the flow structure. A comprehensive review of DNS of bubbly flows is presented in Tryggvason et al. (2006), Tryggvason et al. (2013). Because of the increased complexity of two-phase flow compared to the standard single-phase flow, the first simulations focused on laminar (or pseudo-turbulent) flows.

The first DNS of a turbulent bubbly channel flow with explicit tracking of deformable bubbles has been performed by Kanai and Miyata (2001). A few studies on bubbles and turbulence interaction have followed (*e. g.*, Kawamura and Kodama, 2002; Lu et al., 2005; Lu et al., 2006). In particular, the interest in fully-resolving the bubbles' deformations has been stated by Tryggvason et al. (2006) who demonstrated that the bubbles deformation strongly affects the drag coefficient and the lift force, hence resulting in very different void fraction profiles depending on the value of the Eötvös number $\text{Eo} = \rho g D_b^2/\sigma$ where g is the acceleration due to gravity, D_b is the bubble diameter and σ is the surface tension. Besides, the effect of the direction of gravity has been studied on air/water upward and downward flows (Lu et al., 2006; Lu and Tryggvason, 2008). More recently, Dabiri and Tryggvason (2015) have moved towards the study of convective heat transfer in turbulent bubbly up-flows. They studied the effect of multiphase fluid dynamics on heat transfers, neglecting phase-change and coalescence. The simulated flows reached a Reynolds number of $\text{Re}_\tau = 280$, with up to 84 bubbles. Lately, Tryggvason and Lu (2015) have also shown interest in bubbles of different sizes rising upward in turbulent channel flow. The main characteristics of some of those studies are summarised in Table 1.

To our knowledge, no DNS of high-pressure steam-water turbulent bubbly flow has been achieved yet. The present study is a novelty because we have simulated an upward bubbly flow with 10% void fraction in Pressurised Water Reactor (PWR) conditions, for a friction Reynolds number of 180. The void fraction of 10% is a great improvement compared to the existing literature, even though the achievable Reynolds number is still too low compared to most industrial applications.

3. Numerical method and computational setup

Here, we simulate the rise of buoyant bubbles in turbulent upflow for pressurized steam/water conditions. We first describe the governing equations and the Front-Tracking method used to simulate the flow (Section 3.1). Then, the test-case and the computational domain are described (Section 3.2). Finally, elements of validation on single-phase turbulent flow are given (Section 3.3).

3.1. Governing equations and numerical method

A finite-difference method with Front-Tracking is used to perform the numerical simulations. The "one-fluid" Navier–Stokes equation (Kataoka, 1986; Bunner and Tryggvason, 2003)

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla P + \rho \mathbf{g} + \nabla \cdot \left[\mu\left(\nabla \mathbf{u} + \nabla^T \mathbf{u}\right)\right] + \sigma \kappa \mathbf{n}_v \delta^i \quad (1)$$

is solved over the whole domain, including both the bubbles and the liquid. Here, $\mathbf{u}$ is the velocity vector, P is the pressure, ρ and μ are the discontinuous density and viscosity fields respectively