Nuclear Engineering and Design 300 (2016) 135–148

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Lattice Boltzmann simulation of flow across a staggered tube bundle array

A. Tiftikçi, C. Kocar*

Department of Nuclear Engineering, Hacettepe University, Beytepe, 06800 Ankara, Turkey

HIGHLIGHTS

- Large eddy simulation of the cross-flow in a staggered tube bundle array in 3D was made.
- LBM and FVM are used separately as numerical solvers and the results of each method compared with experimental data.
- Effect of lattice model is studied for tube bundle flow.
- Filter size effects, mesh size effects are studied for VLES turbulence model.

ARTICLE INFO

Article history:
Received 21 May 2015
Received in revised form 4 December 2015
Accepted 23 January 2016
Available online 11 February 2016

K. Thermal hydraulics

ABSTRACT

The decision on the magnitude of the grid size is a crucial problem in large eddy simulations. Finer mesh requires excessive memory and causes long simulation time. Large eddy simulation model becomes inefficient when the extent of the flow geometry to be simulated with the lattice-Boltzmann method is large. Thus, in this study, it is proposed to investigate the capabilities of three turbulence models, namely, very large eddy simulation, Van Driest and Smagorinsky–Lilly. As a test case, a staggered tube bundle flow experiment is used for the validation and comparison purposes. Sensitivity analyses (including mesh and filter size) have been made. Furthermore, the effect of lattice model is investigated and it is showed that the D3Q27 and D3Q19 models do not differ significantly in lattice-Boltzmann method for this type of flow. The results of turbulence model comparisons for staggered tube bundle flow showed that very large eddy simulation is superior at low resolution. This paper might be considered as a good validation of the lattice-Boltzmann method. In turbulent flow conditions, the code successfully captures the velocity and stress profiles even if the flow is quite complicated.

1. Introduction

The development of computer technology and the improvement of numerical methods encourage the simulation of turbulent flows. The modelling of turbulent flow is different from and more sophisticated than laminar flow. Therefore, many turbulence models have been developed and investigated so far. Direct numerical simulation (DNS) is still very expensive even if powerful processors are developed. In DNS, all turbulent scales must be resolved from smallest Kolmogorov scale to the integral length scale. This necessitates a high lattice resolution for turbulent flow simulations. Thus, instead of DNS, large eddy simulation (LES) is generally preferred for turbulence problems. In LES, large eddies interact with and extract energy from the mean flow. These eddies are simulated directly, and the smaller ones (the size smaller than the grid size) are modelled with an appropriate assumption.

The decision on the magnitude of the grid size is a crucial problem in LES. The finer mesh requires extensive memory and causes long simulation time. The coarser mesh may give incorrect results. Thus, the LES model may be inappropriate when the size of the flow geometry is large. Recently, very large eddy simulation (VLES) turbulence model has been introduced to reduce the computational cost of LES. The main idea in this approach is to combine the unique advantages of Reynolds averaged Navier–Stokes equations (RANS) and LES. In this model, larger parts of turbulent fluctuations are filtered. A more comprehensive sub-filter model is needed to eliminate the accuracy lost in filtering concept. The standard $k-\varepsilon$ model is used in sub-grid scale modelling. This allows the usage (with an acceptable accuracy and robustness) a coarser grid than LES.

The main scope in this study is to investigate the capability of VLES for simulating the turbulent flow even if the resolution is low.

* Corresponding author. Tel.: +90 3122977300; fax: +90 3122972122.
E-mail address: ckocar@hacettepe.edu.tr (C. Kocar).

http://dx.doi.org/10.1016/j.nucengdes.2016.01.020

The simulation time and accuracy reached are compared. To do this, three different turbulence models Smagorinsky–Lilly (LES), Van-Driest (LES) and VLES are used to simulate the flow in staggered tube bundle array. VLES and Van-Driest turbulence models are implemented to the lattice-Boltzmann framework. The results obtained from each method are compared with the experimental data.

The experiments performed by Simonin and Barcouda (1986, 1988) are one of the turbulence challenge problem for heat exchanger design which is an attractive test example offered by ERCOFTAC-IAHR (1983, 1994). The case is a two-dimensional isothermal flow across a staggered tube bundle array. Velocity and Reynolds stress profiles along specified lines are measured. Many researchers studied on this problem using different turbulence models and different geometrical modelling of the system. Watterson et al. (1999) used pressure-based finite volume algorithm for solving Reynolds averaged Navier–Stokes equations and their computational domain consists of all seven rows of tubes with one pitch length width in transverse direction. Rollet-Miet et al. (1999) used LES finite element code and presented constant and dynamic Smagorinsky sub-grid model results for fully periodic 3D geometry. Benhamadouche and Laurence (2003) used LES and transient Reynolds stress transport model (RSTM) in 2D and 3D for periodic computational domain. They compared their results with DNS and experimental results. Hassan and Barsamian (2004) simulated a part of full bundle geometry in LES using curvilinear coordinates. Labois and Lakehal (2011) used a new turbulence sub-grid scale model as VLES and compared their results with LES for fully periodic unit cell of staggered tube bundle problem. By considering all discussions and conclusions made by the authors, the resulting consensus is that the calculated Reynolds stresses do not match the experimental data.

In recent years, the lattice-Boltzmann method (LBM) rapidly increases its popularity as an alternative method to the traditional Computational Fluid Dynamics (CFD) methods. LBM does not solve Navier–Stokes equations; instead discrete Boltzmann equation is solved explicitly to simulate the fluid flow with collision model Bhatnagar–Gross–Krook (BGK) introduced by Bhatnagar et al. (1954). Collision and streaming processes of fluid particles evaluate the macroscopic behaviour of the system such as density and velocity. The detailed information about the formulation of LBM is given in the next section.

The remaining part of this paper is organized as follows: In Section 2, the general information about lattice-Boltzmann method with D3Q19 and D3Q27 lattice models is given. Also, the numerical methods adopted in this study are described. LES (Smagorinsky–Lilly and Van-Driest) and VLES sub-grid-scale turbulence models are mentioned for lattice-Boltzmann numerical solvers. In Section 4 and 5, LES (with ANSYS Fluent 13.0 (ANSYS, 2010) and LBM) and VLES computations of flow in a staggered tube bundle array are presented and compared with the experimental data. The results are analyzed, and the important findings obtained from this study are discussed. Finally, in Section 6, the summary and conclusions of this study are presented.

2. Lattice-Boltzmann method with turbulence

The LBM is a new and an alternative approach for simulating fluid flows. In LBM, the continuous fluid flow is decomposed into pockets of fluid particles. The fluid particles may stay at rest or move to one of neighbouring nodes. D3Q19 model is most widely used lattice model for 3D simulations of laminar flow problems. However, turbulence simulations with D3Q19 model could not give reasonable results to the DNS data (Eggels et al., 1994). In Kang and Hassan's (2013) turbulent circular pipe flow simulations, they showed that the D3Q27 lattice model could achieve the rotational invariance for long time averaged turbulence statistics and generates the results comparable to DNS data, while the D3Q19 lattice model breaks the rotational invariance and produces unreasonable data. Thus, in this study, the effect of lattice model on the simulation results is investigated. To do this, fully periodic flow geometry with resolution $D/50$ is selected for comparison.

In D3Q27 model (see Fig. 1), the discrete lattice-Boltzmann equation is in the form of,

$$f_i(x+e_i\Delta t, t+\Delta t) - f_i(x,t) = \Omega(f_i) \quad (1)$$

where $f_i(x,t)$ is the distribution function for particles with velocity e_i at position x and time t, Δt is the lattice time interval and $\Omega(f_i)$ is the collision operator which is defined as,

$$\Omega(f_i) = -\frac{1}{\tau}\left(f_i(x,t) - f_i^{eq}(x,t)\right) \quad (2)$$

where τ is the relaxation time.

The equilibrium function $f_i^{eq}(x,t)$ is written as,

$$f_i^{eq}(x,t) = w_i\rho(x)\left(1+\frac{3}{c^2}e_i\cdot u+\frac{9}{2c^2}(e_i\cdot u)^2-\frac{3}{2c^2}u^2\right) \quad (3)$$

where the weights are

$$w_i = \begin{cases} 8/27 & i=0 \\ 2/27 & i=1,\ldots,6 \\ 1/216 & i=7,\ldots,14 \\ 1/54 & i=15,\ldots,26 \end{cases}$$

here $\rho(x)$ is the space dependent fluid density and the lattice speed is $c=\Delta x/\Delta t$, where Δx is the grid size and Δt is the time step size respectively.

In D3Q27 model, lattice vectors are specified as;

$$e_i = \begin{cases} (0,0,0) & & & i=0 \\ (\pm1,0,0), & (0,\pm1,0), & (0,0,\pm1) & i=1,\ldots,6 \\ (\pm1,\pm1,0), & (\pm1,0,\pm1), & (0,\pm1,\pm1) & i=7,\ldots,18 \\ (\pm1,\pm1,\pm1) & & & i=9,\ldots,26 \end{cases}$$

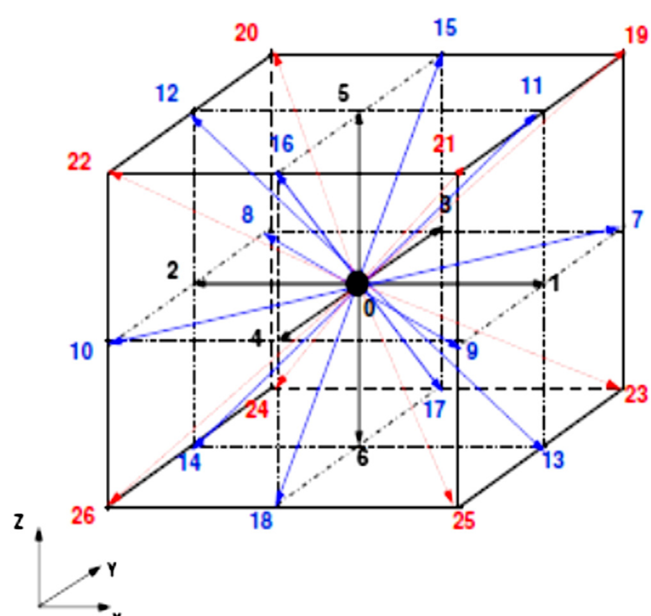

Fig. 1. D3Q27 x, y and z velocity components.