International Journal of Heat and Fluid Flow 49 (2014) 69–79

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

A comparison and assessment of approaches for modelling flow over in-line tube banks

Hector Iacovides *, Brian Launder, Alastair West [1]

Turbulence Mechanics Group, School of MACE, The University of Manchester, UK

ARTICLE INFO

Article history:
Available online 28 June 2014

Keywords:
In-line tube banks
Turbulent flow
Turbulence modelling
Large-eddy simulation
Convective heat transfer

ABSTRACT

The paper reports experiences from applying alternative strategies for modelling turbulent flow and local heat-transfer coefficients around in-line tube banks. The motivation is the simulation of conditions in the closely packed cross-flow heat exchangers used in advanced gas-cooled nuclear reactors (AGRs). The main objective is the flow simulation in large-scale tube banks with confining walls. The suitability and accuracy of wall-resolved large-eddy simulation (LES) and Unsteady Reynolds-Averaged Navier–Stokes (URANS) approaches are examined for generic, square, in-line tube banks, where experimental data are limited but available. Within the latter approach, both eddy-viscosity and Reynolds-stress-transport models have been tested. The assumption of flow periodicity in all three directions is investigated by varying the domain size. It is found that the path taken by the fluid through the tube-bank configuration differs according to the treatment of turbulence and whether the flow is treated as two- or three-dimensional. Finally, the important effect of confining walls has been examined by making direct comparison with the experiments of the complete test rig of Aiba et al. (1982).

1. Introduction

This research is motivated by the need to understand the flow and thermal processes in cross-flow heat exchangers of advanced gas cooled (AGR) nuclear reactors. Engineering applications of cross-flow tube banks are abundant. Such configurations achieve high heat transfer with relatively low manufacturing complexity, making them attractive heat exchangers for use in fossil-fuel and nuclear power plants. Reliable prediction of the flow and heat-transfer characteristics of such tube-bank flows is therefore essential for heat-exchanger design and life-time management. Such heat exchangers may consist of arrays of hundreds or even thousands of tubes, through which a fluid passes (or heat-releasing solid is contained) while a second fluid is blown normal to them, the overall purpose being to promote heat exchange between the two fluids. Detailed testing on such systems, both experimental and computational, is largely done on much smaller systems, typically consisting of clusters comprising from four to a few tens of tubes, the hope being that the data emerging will be representative of those in the full-scale plant.

Experiments on widely-spaced in-line and staggered tube banks have been carried out *inter alia* by Ishigai et al. (1973) where several distinct flow patterns were observed. Costs with tightly packed tube bundles are lower, however, and extensive data have been reported on close-packed *staggered* tube banks. There are, however, few experiments of closely-spaced in-line tube banks (Iwaki et al., 2004) and even fewer providing data of local heat transfer (Zukauskas, 1989, reporting his team's results and Aiba et al., 1982, providing notable exceptions).

Large-eddy simulations (LES) of closely-spaced, quasi-infinite square in-line tube banks of various pitch-to-diameter ratios, P/D, have been conducted in Manchester by Benhamadouche (2006) and Afgan (2007). The latter examined the behaviour for four values of P/D. At $P/D = 1.75$ the mean-flow streamlines exhibited a conventional straight-through, symmetric behaviour with recirculating eddies immediately behind the upper and lower halves of the cylinders. For a P/D of 1.6 and, sometimes for 1.5 too, an alternating behaviour was found where a single large separated eddy was formed on the downstream side of the cylinder, but its location alternated in successive rows from the upper to the lower side of the cylinder. At other times, for $P/D = 1.5$ and always for $P/D = 1.2$, the mean flow's passage through the bank was diagonal, the angle of departure of the flow from the horizontal increasing as P/D was reduced. It appeared that the mean flow preferred to travel in a diagonal path through the domain, thereby minimising

---

* Corresponding author. Tel.: +44 0161 306 3709; fax: +44 0161 306 3723.
E-mail address: h.iacovides@manchester.ac.uk (H. Iacovides).

[1] Present address: CD-adapco, 200 Shepherds Bush Road, London W6 7NL, UK.

http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.05.011
0142-727X/© 2014 Elsevier Inc. All rights reserved.

the pressure drop (or maximising the flow rate for a given pressure drop).

Such a diagonal flow path has seldom been seen in experiments in the small arrays normally adopted because then the confining wind-tunnel walls restrict cross-flow motion. However, Jones et al. (1978) reported cross-flow drift in their 22 × 22 tube test section while Aiba et al. (1982) noted, for $P/D = 1.2$ with only 4 tubes in the cross-flow direction, that "it is very clear that the flow through the tube bank deflects as a whole".

The numerical simulation of large tube-bank arrays can be greatly simplified (or, at least made manageable) by considering, as in the above-cited studies, a small subset of the complete bank and assuming flow periodicity at the boundaries of the sub-domain considered: what were referred to above as 'quasi-infinite'. These cyclical boundaries are placed at user-chosen locations where the flow is judged to repeat itself. But, one may reasonably ask, does such a simulation *really* give an adequate account of the flow and thermal behaviour within the region unaffected by\ walls? Equally, what complexity do the bounding walls that are present in an actual heat exchanger bring to the flow behaviour?

It is the aim of the present paper to explore both the above questions and, in the process, reach at least tentative conclusions on best modelling practices. We first examine the repeating flow that is generally supposed to arise deep within the tube bank to explore whether or not, at the LES level, one is able to confirm recently reported results regarding the diagonal flow behaviour and its dependence on the pitch-to-diameter ratio of the bank. Thereafter, we proceed to examine how well unsteady RANS (hereafter 'URANS') can mimic these LES results (for the latter requires at least an order of magnitude more computational resource than the former). To address the question of the effect of the confining boundaries on the flow, simulations are made of the complete tube bank used in the Aiba et al. (1982) experimental study, which provides the second stage of validation. While the Aiba tube bank is far smaller than that of full-scale industrial tube banks, such as those in AGR nuclear reactors, the conclusions reached about the impact of the enclosing walls on the overall flow pattern also have some relevance to larger-scale applications.

This investigation adopts not our in-house software and preferred turbulence modelling practices but the freely available industrial program, *Code_Saturne* developed by Électricité de France (EdF) that is becoming widely used within the European heat-transfer community.

2. Computational and physical models

2.1. Discretisation practices and boundary conditions

Both LES and URANS approaches have, as noted above, employed the finite-volume code, *Code_Saturne* (Archambeau et al., 2004) with a collocated grid. It was decided to use this freely available and versatile software although it does not offer some of the more advanced modelling practices incorporated in our in-house code, STREAM (Lien and Leschziner, 1994; Craft et al., 2004b). However, a few computations were also made with the latter code (West, 2012) which broadly confirmed the conclusions reached with *Code_Saturne*. The velocity–pressure coupling is achieved by a predictor/corrector method using the SIMPLEC algorithm where the momentum equations are solved sequentially. The Poisson equation for the pressure-correction field is solved using a conjugate-gradient method and a standard pressure-gradient interpolation to avoid oscillations. As spatial and temporal discretisation are second order (central-difference and Crank-Nicolson interpolations respectively), the time step was kept sufficiently small to ensure the maximum Courant number was below unity.

To explore the case of fully-developed flow, periodic boundaries are imposed in all three Cartesian directions placed at distances of L_x, L_y and L_z, see Fig. 1a where L_x and L_y are adjusted to obtain the desired pitch:diameter ($P{:}D$) ratio. Fig. 1b shows a portion of the computational grid employed for the standard (Case 1) grid. Block-structured grids gave better control of the number of cells and a more effective resolution of the near-wall regions than the alternative strategies available. As flow periodicity is used, a constant mass flow rate is imposed to obtain the desired bulk velocity by specifying an explicit self-correcting mean pressure gradient at every time step. Previous periodic calculations of in-line tube banks (Afgan, 2007; Beale and Spalding, 1999; Benhamadouche, 2006) have found a 2 × 2 tube domain to be sufficient to capture the unsteady flow characteristics and mean quantities of interest. While Benhamadouche et al. (2005) also tested a larger 4 × 4 tube domain for an in-line bank of $P/D = 1.44$, the same flow pattern was predicted in each case and no differences in mean quantities were reported.

A uniform heat flux is prescribed on the tube surfaces. To maintain a fixed bulk temperature as iterations proceed, the periodic inlet temperature distribution is rescaled using a bulk correction corresponding to the total amount of energy added to the domain. Thermo-physical fluid properties are assumed to be constant.

For the URANS computations, grid sensitivity studies were first performed for both high-Re (i.e. used with wall functions) and low-Re (integration to the wall) grids, full details of which are given in West (2012). In the immediate vicinity of the tubes the grid is cylindrical polar and distances below are expressed in terms of the wall-normal coordinate n and the circumferential coordinate s. For high-Re grids most near-wall nodes lie within the turbulent boundary layer ($30 < n^+ < 200$), whereas for the low-Re grids a dimensionless wall distance of less than unity was maintained ($n^+ < 1$). For the low-Re grids a radial-expansion factor of 1.1 was used from the wall to sufficiently resolve the near-wall sublayer, ensuring at least 25 nodes are located over the viscous and buffer regions. The finest low-Re grid was used as a starting point for the LES grid sensitivity studies. The optimum numbers of grid cells for the LES and other grid parameters are given in Table 1. The grid parameters for the LES resolution and domain-size studied are summarised in Table 2. A factor of 1.1 is used for the radial grid-expansion from the tube walls. Around the tube surface 160 cells were used except 256 cells for Case 4. Fig. 2 shows the wall-adjacent cell size around the central cylinder in wall units for Case 1. The centre of the wall-adjacent cell is located at $n^+ = \Delta n^+/2$ which was approximately equal to 0.25 over most of the cylinder wall. The spanwise resolution was initially chosen (Case 1) in order that, *in the central region*, cell dimensions were comparable with those in the streamwise and cross-wise direction (i.e. they were as close as possible to a regular hexahedron or a cube). Piomelli and Chasnov (1996) recommend $n^+ < 2$, $\Delta s^+ = 50\text{–}150$, $\Delta z^+ = 15\text{–}40$ for a wall-resolving LES. Their proposals imply that Case 1 needed better resolution in the z-direction to resolve fully the near-wall structures. Case 4, with over three times as many cells as Case 1, meets these recommendations with a mean Δz^+ of around 30. Case 3, with 130 spanwise cells also meets the recommendation.

2.2. Turbulence modelling

The turbulence models used for the URANS calculations are listed in Table 3 (these amount to the majority of those available within *Code_Saturne*). A few remarks on each model are provided below while a complete mathematical statement is given in the cited references for each scheme.

2.2.1. Linear production (LP) k–ε model (Guimet and Laurence, 2002)

This variant on the usual k–ε eddy-viscosity model is designed to remove the well-known weakness of that model (and, indeed,