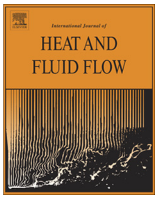
International Journal of Heat and Fluid Flow xxx (2014) xxx–xxx

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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



Hybrid simulations of the near field of a split-vane spacer grid in a rod bundle

Dongil Chang, Stavros Tavoularis *

Department of Mechanical Engineering, University of Ottawa, Ottawa, Ontario K1N 6N5, Canada

ARTICLE INFO

Article history:
Available online xxxx

Keywords:
LES
Scale adaptive simulation
Rod bundle
Vane grid spacer
Vortices
Turbulent mixing

ABSTRACT

Three-dimensional, unsteady simulations of isothermal turbulent flows in a rod bundle with a split-vane spacer grid have been performed using a segregated turbulence model, which is a combination of scale adaptive simulations and large eddy simulations. These simulations have been conducted within the framework of an international blind CFD benchmark exercise. A finer mesh than that submitted to the benchmark exercise was used for the present study, which improved the agreement of the turbulence predictions with the measurements. For the first time, several vortices were identified in the vicinity of the vanes. The strongest among these vortices, which was a central vortex in the core of each subchannel, was generated by the vane pair in each subchannel. Each vane also created a strong stream across the gap between two rods and towards an adjacent subchannel. Axial profiles of turbulent kinetic energy in each subchannel core exhibited two peaks, a low peak in the near-vane zone and a larger peak one hydraulic diameter downstream of the vanes.

1. Introduction

The cores of pressurized water nuclear power reactors (PWR) contain a number of fuel rod assemblies, which are also known as fuel rod bundles. For example, the Westinghouse AP1000 light water reactor (LWR) core contains a matrix of 157 fuel assemblies, each comprising a 17 × 17, square-type, fuel rod array with a length of 4.3 m (Xuesong, 2010). Each LWR fuel assembly has spacer grids installed at regular intervals, which keep the fuel elements (rods) in place and prevent their excessive vibration. In addition to this function, spacer grids are designed to enhance turbulent mixing within the surrounding subchannels and thus increase the heat transfer rate between the fuel elements and the coolant. This reduces the chance of local boiling and widens the safety margin of the nuclear power plant during normal operation. A common practice is to fit the downstream end of each spacer grid with mixing vanes, which generate vortices to promote mixing between the hot fluid near the rod surfaces and the cooler fluid in the centre of the subchannel. Spacer grids and the attached vanes also generate additional hydraulic resistance to the flow, which necessitates additional pumping power to drive the coolant through the reactor core. For this reason, the LWR spacer grid design aims at producing the strongest possible mixing and the lowest possible flow resistance. To achieve an optimal vane design, one needs to optimize the vane shape, size and orientation. This process may require many iterations, which would be difficult and expensive to perform by experimental means alone. Computational fluid dynamics (CFD) analysis has emerged as a cost- and time-effective alternative approach for evaluating the thermal–hydraulic performance of nuclear reactor components, at least in the conceptual and design stages. To be credible, CFD predictions need to be validated, to the greatest degree possible, against measurements at appropriate conditions. In the case of nuclear reactors, it is essentially impossible to measure the velocity and temperature variations within the reactor core under normal operating conditions, and even more so under conditions of nuclear accidents. Therefore, CFD simulations for nuclear reactor components are by necessity validated against laboratory experiments, which are usually conducted under relatively low pressures and temperatures and at Reynolds numbers that are typically an order of magnitude lower than those in operating reactors, which is typically 5×10^5.

Two types of measurements of velocity and turbulence characteristics in rod bundles with mixing vanes relevant to LWR cores are available: point measurements, obtained by laser Doppler velocimetry (LDV; Shen et al., 1991; Yang and Chung, 1998; Chang et al., 2008) and measurement maps on planes normal to the flow, obtained by particle image velocimetry (PIV; Smith

---

* Corresponding author. Address: Department of Mechanical Engineering, University of Ottawa, 161 Louis Pasteur, Ottawa, Ontario K1N 6N5, Canada. Tel.: +1 613 562 5800x6271; fax: +1 613 562 5177.

E-mail address: stavros.tavoularis@uottawa.ca (S. Tavoularis).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.07.005
0142-727X/© 2014 Elsevier Inc. All rights reserved.

Please cite this article in press as: Chang, D., Tavoularis, S. Hybrid simulations of the near field of a split-vane spacer grid in a rod bundle. Int. J. Heat Fluid Flow (2014), http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.07.005

et al., 2002; McClusky et al., 2002, 2003). These studies (e.g., Shen et al., 1991; McClusky et al., 2002, 2003; Chang et al., 2008) have demonstrated that the vanes generated swirling flows within the rod-bundle subchannels and/or cross flows in the gap regions between the rods, whose patterns and strengths varied widely depending on the vane design. It has also been demonstrated that, compared to the reference level in bare rod bundles, vanes significantly enhanced the turbulence intensity, which had a peak value downstream of vanes and then decreased monotonically to the reference level in the bare rod bundle.

Most earlier CFD analyses of flows in rod bundles with grid spacers were solutions of the RANS (Reynolds-averaged Navier–Stokes) equations with mainly isotropic two-equation turbulence models (Smith et al., 2002; Uchida et al., 2009; Conner et al., 2010; Gandhir and Hassan, 2011; Navarro and Santos, 2011). Smith et al. (2002) and Uchida et al. (2009) predicted peak velocities on cross-planes near the vanes that were significantly higher than the experimental values; this discrepancy may be attributed to the inability of isotropic turbulence models to describe the complex flow structure near the vanes. Lee and Choi (2007) also used the RANS approach to simulate flows in rod bundles with spacers fitted with two types of mixing vanes, but employed a Reynolds stress model (RSM), which resolved individual Reynolds stresses; these authors demonstrated that the turbulence was strongly anisotropic near the vanes, but did not present comparisons of their predictions with measurements.

In an effort to assess the accuracy of CFD simulations and to establish best practice guidelines for the nuclear industry, the Committee on the Safety of Nuclear Installations (CSNI) of the Nuclear Energy Agency (NEA), affiliated with the Organization for Economic Cooperation and Development (OECD), organized the Second International Benchmark Exercise (IBE-2) on turbulent mixing in a rod bundle with spacer grids, which took place during the period between April 2011 and May 2012. The Korea Atomic Energy Research Institute (KAERI) provided details of the test geometry and conditions of the tests performed in the 5 × 5 rod bundle MATiS-H (Measurements & Analysis of Turbulence in Subchannels – Horizontal) facility (OECD/NEA, 2012). Two types of vanes, split vanes and swirl vanes, were used for this benchmark. Summaries of the results of this exercise have been presented in a recent article by Lee et al. (in press). The rankings of our submission will be discussed in Section 6.

Since the completion of the IBE-2 exercise, two CFD studies of the MATiS-H split-vane configuration have been published. Capone et al. (2013) conducted a series of unsteady simulations using a mesh with 62 M cells and three different turbulence models: LES with the dynamic Smagorinsky subgrid model (DSM), LES with the wall adapting local eddy viscosity subgrid model (WALE), and URANS (Unsteady RANS) with the SSG (Speziale–Sarkar–Gatski) RSM model. They compared their predictions of spanwise turbulent stress and turbulence kinetic energy against measured data and concluded that LES with WALE performed best, although even their best predictions were far from the measurements at two planes near the outlet. Podila et al. (2014) used URANS with RSM and a mesh of 15 M cells. Their mean velocity predictions were in fair agreement with measurements but the turbulent stresses and the circulation were underpredicted.

The present work is an extension of the simulations performed during our participation in IBE-2. This paper presents results of simulations with a refined mesh, which were not submitted in time to the organizers due to time constraints. Our objective is to predict the mean velocity and turbulent distributions in the MATiS-H rod bundle with grid spacers fitted with split vanes and, in particular, to investigate the detailed flow characteristics near the vanes. In this analysis, we used a segregated hybrid approach, consisting of a combination of Scale Adaptive Simulations (SAS) and Large Eddy Simulations (LES), in an effort to achieve the accuracy of LES at a computational cost that was lower than that of full LES. The specific results of this article do not necessarily apply to commercial grid spacers because the presently analyzed geometry was intended for use in the OECD benchmark exercise and was not optimized for installation in operating nuclear reactors. Nevertheless, the present study fills a gap in the literature on flows in rod bundles with grid spacers, as it describes in detail a computational procedure that is capable of illuminating the mechanisms by which mixing vanes generate vortices and cross flows to enhance turbulent mixing, and so can be applied for the development of optimal grid spacers for industrial use.

2. Computational procedures

The rod bundle in the MATiS-H test rig was contained in a square duct with a side equal to 170.0 mm. The rods had an outer diameter of 25.4 mm, which was 2.6 times larger than that in a typical LWR, and a length of 3863.0 mm. The duct also had a short section upstream of the rod bundle, which contained a flow straightener, with the objective to produce a nearly uniform velocity across the duct. A second flow straightener was inserted in the section containing the rod bundle, at a distance 2430.0 mm upstream of the spacer grid (S/G). Velocity measurements in the rod bundle flow were only conducted at a cross section located 5.0 mm upstream of the downstream end of the rod bundle. The measuring system was a two-dimensional laser-Doppler anemometer (LDA).

Simulating the flow in the entire MATiS-H test rig would require excessive computational resources (RAM memory); for this reason, our computational domain included only the part of the test section that was deemed to be important for our purposes. We divided the computational domain to two distinct parts: the first part included the bare-rod bundle section between the second flow straightener and the spacer grid of the experimental section, whereas the second part included the rod bundle section with the spacer grid. The objective of computations in the first part was to produce inlet boundary conditions for the main computations, which were performed in the second part.

The computational domain included the entire cross section of the rod bundle duct rather than a section of it, in anticipation of the spatial non-periodicity of the flow downstream of the split vane spacer grid. Fig. 1a shows a cross-section of the bare 5 × 5 rod bundle in a square duct, which had a width $h = 170.0$ mm. Each rod had an outer diameter $D = 25.4$ mm and the hydraulic diameter of the rod bundle cross-section was $D_h = 24.3$ mm. The rod and wall pitches were $P = 33.1$ mm and $W = 31.5$ mm, respectively, which correspond to a rod pitch-to-diameter ratio $P/D = 1.30$ and a wall-pitch-to-diameter ratio $W/D = 1.24$. The same figure also illustrates the spacer grid with the split-vanes, which were designed to generate vortices that enhance turbulent mixing, and the grid buttons, whose purpose is to keep the fuel rods in place. A section of the rod bundle, on which we will focus the discussion, is shown in Fig. 1b. The flow area in this section consists of an inner, an intermediate and a wall subchannel. Each subchannel except the wall subchannel contains two vanes, inclined in opposite directions to form angles of ±30 deg with the streamwise direction. Pairs of vanes in neighbouring subchannels are arranged in alternating orientations; for example, the roots of vanes in the inner subchannel are aligned with the y-axis, whereas the roots of vanes in the intermediate subchannel are aligned with the x-axis. Comparisons between predictions and measurements will be mainly reported along a line with $y/P = 0.5$.

Fig. 2a shows a side view of the computational geometry. The length of the entire geometry was approximately $27.1D_h$, which

Please cite this article in press as: Chang, D., Tavoularis, S. Hybrid simulations of the near field of a split-vane spacer grid in a rod bundle. Int. J. Heat Fluid Flow (2014), http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.07.005