Applied Thermal Engineering 40 (2012) 389–396

Contents lists available at SciVerse ScienceDirect

Applied Thermal Engineering

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

CFD evaluation of turbulence models for flow simulation of the fuel rod bundle with a spacer assembly

C.C. Liu [a], Y.M. Ferng [a,b,*], C.K. Shih [a,b]

[a] Institute of Nuclear Engineering and Science, National Tsing Hua University, 101, Sec. 2, Kuang-Fu Rd., Hsinchu 30013, Taiwan
[b] Department of Engineering and System Science, National Tsing Hua University, Hsinchu, Taiwan

ARTICLE INFO

Article history:
Received 11 October 2011
Accepted 13 February 2012
Available online 20 February 2012

Keywords:
Turbulence modeling
CFD
Rod bundle

ABSTRACT

Computational fluid dynamics (CFD) is increasingly being used in nuclear reactor safety (NRS) analyses to describe safety-relevant phenomena occurring in the reactor coolant system in greater detail. This paper presents the results of numerical issues such as mesh refinement, wall treatment and appropriate definition of boundary conditions, which exert great influence on the results of a CFD simulation. A high quality computational mesh was used to investigate the choice of turbulence model appropriate for the complex swirling flow in the rod bundle subchannels. The performance of various turbulence models are evaluated by calculation of the Nusselt number distribution in a fuel bundle. Comparison between numerical and experimental results of lateral and axial distributions for the Nusselt number obtained via turbulence model without near-wall functions is not sufficiently good, while agreement is found between the computational simulation of the realizable k–epsilon model with near-wall functions and the experimental measurements, for locations close to the support grid. As a result of this study, we have been able to determine the most appropriate turbulence models and the best enhanced wall treatment for modeling reactor coolant systems.

1. Introduction

Two tools have been widely used in experimental measurement: Laser Doppler Velocimetry (LDV) and Particle Image Velocimetry (PIV). These methods allow local velocities to be measured between the subchannels of the rod bundle. Despite the great improvement recently attained by these techniques, the experimental investigations are still expensive and require a relatively long time to perform. On the other hand, in theoretical evaluations such as computational fluid dynamics (CFD) analysis, results are produced in a relatively short time, especially when the Reynolds-averaged Navier–Stokes (RANS) equations are solved, at much lower costs. Recently, in rod bundle evaluations, the CFD methodology has been used to optimize both the experimental procedures and the geometric issues. In recently performed CFD investigations using the RANS analysis, different numerical simulation procedures and results analysis methodologies have been used to evaluate the configuration of the flow characteristics throughout rod bundles. This demonstrates that these methodologies are still a subject of debate.

There have been several studies on flow mixing and heat transfer enhancement caused by a mixing vane spacer grid in rod bundle geometry. Navarro et al. [1] used the k–ε model that presents results of flow simulations performed with the CFD code in a PWR 5×5 rod bundle segment with a split-vane spacer grid. M.E. Conner et al. [2] conducted experiments to validate the CFD methodology for the single-phase flow conditions in PWR (Pressurized Water Reactor) fuel assemblies. Holloway et al. [3] showed that there is a great variation of heat transfer distribution along a fuel rod due to the spacer grid type. A series of four-subchannel CFD simulations to analyze the heat transfer enhancement in a fully heated rod bundle with vane spacers were performed by In et al. [4]. Lee and Choi [5] also used the Reynolds stress model (RSM) to compare the performance of spacer vane designs between the small scale vortex flow (SSVF) mixing vane and the large scale vortex flow (LSVF) mixing vane. By adjusting model coefficients adopted in a quadratic k–ε model by Shih [6], Baglietto and Ninokata [7] had previously shown the promising capability of the RSM turbulence model in a sufficiently accurate anisotropy modeling of the wall shear stress distribution and the velocity field in tight lattice fuel bundles. Hàzi [8] had demonstrated that the RSM could be accurately applied in simulating the rod bundle geometry. Kim and Seo [9,10] presented an optimized mixing vane shape and showed its theoretical effect on the flow structure, using the single-

* Corresponding author. Institute of Nuclear Engineering and Science, National Tsing Hua University, 101, Sec. 2, Kuang-Fu Rd., Hsinchu 30013, Taiwan.
E-mail address: ymferng@ess.nthu.edu.tw (Y.M. Ferng).

1359-4311/$ – see front matter © 2012 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2012.02.027