Annals of Nuclear Energy 41 (2012) 17–25

Contents lists available at SciVerse ScienceDirect

Annals of Nuclear Energy

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

CFD analysis of the effect of rolling motion on the flow distribution at the core inlet

B.H. Yan [a,*], G. Zhang [b], H.Y. Gu [b]

[a] Department of Nuclear Science and Engineering, Naval University of Engineering, Wuhan 430033, China
[b] School of Nuclear Science and Engineering, Shanghai Jiao Tong University, Shanghai 200240, China

ARTICLE INFO

Article history:
Received 12 May 2011
Received in revised form 30 October 2011
Accepted 10 November 2011
Available online 16 December 2011

Keywords:
Rolling motion
Flow distribution
Reynolds number
CFX

ABSTRACT

The flow distribution at the core inlet in rolling motion is investigated with software CFX12.0. The calculation results were in agreement with experimental data in steady state. As the increasing of rolling amplitude and the decreasing of rolling period, the effect of rolling motion on the flow distribution factor and the flowing behavior increases. In rolling motion, the variation of flow distribution factor is not regular. The rolling motion could decrease the minimum flow distribution factor. The effect of rolling motion on the coolant field and flow distribution diminishes with the Reynolds number increasing. The effect of rolling motion on the flow distribution in the case of single loop operation is more significant than that in the case of double loops operation.

1. Introduction

In order to realize the best performance of nuclear reactor core in normal operation condition and keep the structure integrity and reactor safety, a fully understanding of the flowing behavior of coolant flow in the core is very important and necessary. The flowing behavior is dominated by the flow field which is related with the detailed structure and configuration of reactor core (Li and Hu, 2002). Therefore, the investigation and thorough understanding of the flow field and its related flowing behavior in nuclear reactor system, especially in the lower plenum and reactor core, is very necessary. This could not only improve the thermal hydraulic calculation in the reactor but also provide reliable results for the safety analysis of nuclear reactor.

The coolant flow distribution at the reactor core inlet plays an important role in the thermal hydraulic characteristics and safety of the reactor system. Obtaining the correct flow distribution factor at the core inlet is also a fundamental task of nuclear thermal hydraulic design. The flow distributions obtained experimentally and theoretically from the reactor thermal hydraulic models are of vital importance for the validation and optimization of reactor core and lower plenum. Besides that, the flow distributions are also important input parameters of the nuclear thermal hydraulic analysis (Wang et al., 1999).

In the past decade, commercial CFD codes have been applied to nuclear power plant (NPP) real geometry with the aim of examining local thermal hydraulic phenomena such as the safety injection flow in the downcomer (Kwon et al., 2003), turbulence due to the incorporation of a mixing vane (In et al., 2001). Jeong and Han (2008) also analyzed the flow distribution in the downcomer and lower plenum of Korean standard nuclear power plants (KSNPs) with commercial CFD code STAR-CD. In their work, the real geometry is used. Their results provide a clear figure about the flow distribution in the reactor vessel, which is a major safety concern.

In the CFD analysis of flow field and turbulence mixing in reactor pressure vessel, the mesh and turbulence model play important roles in the accuracy of the results. Usually, the calculation results are more satisfactory if better meshes are introduced. However, as the increasing of mesh number, the computational resources increase sharply. Therefore, the mesh generation is usually restricted strictly by an upper limit.

It is well known (ANSYS CFX, 2005) that the standard k–ϵ model shows important weaknesses in predicting:

- Flow impingement and reattachment.
- Swirling and re-circulation flow.
- Flow with strong buoyancy effects and high streamline curvature.
- Turbulence driven secondary flows.

However, the application results of the turbulence models are not absolutely the same. Ikeda et al.'s (2006) results confirmed that predicted lateral velocity using k–ϵ model showed fairy good agreement with the measured velocity in a 5×5 rod bundle. Sofu et al. (2004) have evaluated the Reynolds Averaged Navier Stokes (RANS) models including the standard k–ϵ model, quadratic and cubic k–ϵ models, the renormalization group (RNG) variant, and Reynolds Stress Model (RSM) model. Their results showed that the nonlinear quadratic k–ϵ model is superior to the standard k–ϵ model.

* Corresponding author. Tel.: +86 21 34206277.
E-mail address: binghuoy@163.com (B.H. Yan).

0306-4549/$ - see front matter Crown Copyright © 2011 Published by Elsevier Ltd. All rights reserved.
doi:10.1016/j.anucene.2011.11.004

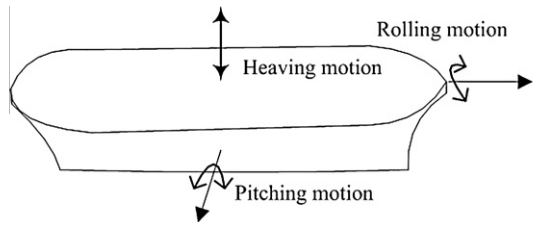

Fig. 1. Schematic of ship motions.

However, the RSM model provides the best agreement with experimental results. Chun et al. (2004) also reported that the RSM model showed excellent performance for complex geometries in spite of very large computing costs. For the VVER mixing test calculations (Rohde et al., 2007), the standard $k–\epsilon$ model with non-equilibrium wall functions and RSM with standard wall functions gave the best agreement. RSM with non-equilibrium wall functions and the SST $k–\omega$ model overestimated the mixing. In general, standard turbulence models implemented in the codes can be used for turbulent mixing calculations. However, for the ROCOM steady state mixing and non-buoyant transient mixing, the results were not very sensitive to turbulence model (the Standard $k–\epsilon$ model and the SST $k–\omega$ model gave similar results) (Rohde et al., 2005). Because of the non-uniformity of the superiority of each turbulence model in the reactor pressure vessel and the limit of mesh refinement, the validation of calculation results with experimental data is very important and indispensable for the CFD analysis in the reactor pressure vessel.

In recent years, there has been a growing interest in the thermal hydraulic analysis of nuclear power system in ships (Panov et al., 1998). Because of the effect of additional force due to ocean environment, the flow distribution in the reactor in ocean environment is different from that in steady state. The main difference from a fluid mechanics point of view between a land-based and ship-based equipment is the influence of sea wave oscillations on the latter. The thermal hydraulic behavior of ship-based equipment is influenced by different motions such as rolling, pitching and heaving motions (Fig. 1). Oscillations change the effective forces acting on the fluid and induce flow fluctuations, which result in a change in momentum, heat and mass transfer characteristics (Pendyala et al., 2008; Tan et al., 2009).

In ocean environment, the flow is affected by the additional force variant with space and time. The experimental results of Du and Zhang (2010) and the theoretical results of Yan et al. (2010a,b,c) indicate that the effects of ocean environment on the flowing behavior and heat transfer characteristics in closed channel (like circular and rectangular tubes) is weak because of the obstruction of channel wall. However, in the reactor pressure vessel, the channel is not closed and the transverse mixing should not be neglected. In this case, the flow field and its relevant flowing behavior may be affected by the additional force caused by ocean environment.

In the present paper, the flow distribution at the core inlet of the scaling apparatus of 600 MW reactor in Qinshan phase II (Li and Hu, 2002; Liu et al., 2003; Wang et al., 1999; Yang et al., 2003; Zhang et al., 2008a,b) in rolling motion is investigated. The calculation results were validated with experimental data in steady state. The effects of several parameters on the flow distribution at the core inlet are also analyzed.

2. Computational domain

2.1. Geometry

In the present work, the studying object is the scaling experimental apparatus of 600 MW reactor in Qinshan phase II (Liu et al., 2003; Wang et al., 1999; Yang et al., 2003; Zhang et al., 2008a,b). The scale of the experimental apparatus and the real reactor pressure vessel is 1:4. The experimental geometry is used in this work (Fig. 2). The detailed description and geometrical parameters are listed by Liu et al. (2003), Wang et al. (1999) and Yang et al. (2003). The whole computational domain is divided into four subdomains: (1) inlet section of reactor pressure vessel and the downcomer, (2) lower plenum and the bottom part of the core (Fig. 3), (3) reactor core, and (4) outlet section of the reactor pressure vessel. The geometrical structure of the computational domain is developed with Solidworks2006.

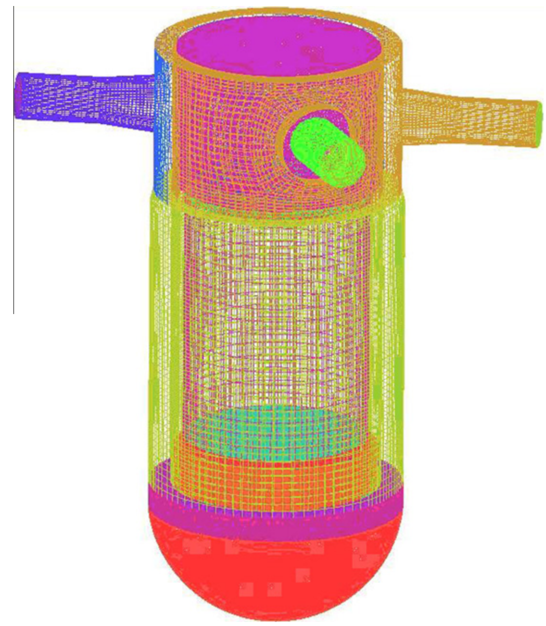

Fig. 2. Computational domain.

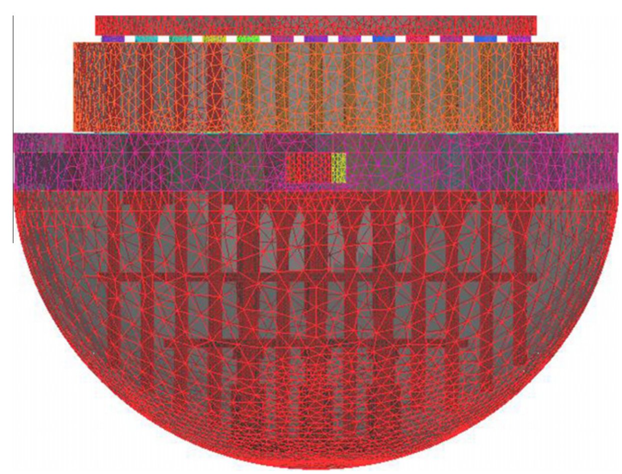

Fig. 3. The structure and mesh generation in lower plenum.

The heat shield in the downcomer, with the small scale and thin panel, is located surrounding the core. It is omitted in the calculation since it imposes little influence on the flow distribution and flow field (Zhang et al., 2008a,b). Therefore, the downcomer is in a regular shape and could be dispersed with structure mesh, which is benefit for improving the calculating speed. In the lower plenum and outlet section of the reactor, since the inner structure and configuration is complex, unstructured mesh is adopted in these two subdomains. The structure meshes are generated in the other two subdomains. Finally, a total of 4,333,487 cells are generated.