Progress in Nuclear Energy 56 (2012) 1–6

Contents lists available at SciVerse ScienceDirect

Progress in Nuclear Energy

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

CFD analysis of the loss coefficient for a 90° bend in rolling motion

B.H. Yan [a,*], H.Y. Gu [b], L. Yu [a]

[a] Department of Nuclear Science and Engineering, Naval University of Engineering, 717 Jiefang Street, Wuhan 430033, China
[b] School of Nuclear Science and Engineering, Shanghai Jiao Tong University, 800 Dong Chuan Road, Shanghai 200240, China

ARTICLE INFO

Article history:
Received 11 June 2011
Received in revised form
1 November 2011
Accepted 10 November 2011

Keywords:
Loss coefficient
90° bend
Rolling motion

ABSTRACT

The local loss coefficient for a 90° bend in rolling motion is investigated with CFD code FLUENT. The calculation results are validated with experimental and theoretical results in steady state. The effect of spanwise and transverse additional forces on the bend loss is significant. The effects of additional forces on the bend loss are mainly embodied in the downstream section. The oscillation of bend loss caused by the spanwise and transverse additional forces is very regular while that caused by velocity oscillation is very irregular. The effect of velocity oscillation on the bend loss is significant in rolling motion with low Reynolds number. But the variation of bend loss coefficient with velocity oscillating period is very limited.

1. Introduction

In ocean environment, 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., 2008a; Tan et al., 2009a, b). Isshiki (1966) was the first scholar to analyze the effect of ocean environment on the thermal hydraulic performance of water cooled reactor. Ishida et al. (1990) investigated the effect of heaving and rolling motions on the thermal hydraulic behavior of marine reactors numerically with a modified RETRAN02 code. Murata et al. (1990, 2002) carried out a series of single-phase natural circulation tests in a model reactor with rolling motion was performed, in order to investigate the natural circulation characteristics of a marine reactor in a stormy weather. The heeling and heaving experiments of small Pressurized Water Reactor (PWR) have been carried out by Ishida and Yoritsune (2002) with the deep sea research reactor (DRX). They also analyzed the operational characteristics of DRX in ocean environment with RETRAN02 code. Pendyala et al. (2008a, b) investigated the effect of axial flow oscillation on the flow and heat transfer in a vertical tube experimentally. Tan et al. (2009a, b) have also finished a series of experiments for studying the single-phase natural circulation flow and heat transfer under rolling motion condition. Yu et al. (2008), Yan et al. (2009, 2010) and Yan and Yu (2009) analyzed the flow and heat transfer characteristics of laminar flow and turbulent flow in ocean environment by establishing mathematical models. Their results indicate that the ocean environment may affect on the thermal hydraulic behavior of nuclear reactor systems.

Since traditional theoretical models and correlations could not predict the flow and heat transfer behavior in ocean environment correctly, most recent works were done for the frictional resistance coefficient and Nusselt number in channels in ocean environment (Pendyala et al., 2008a, b; Tan et al., 2009a, b; Yan et al., 2009, 2010; Yan and Yu, 2009). To the authors' knowledge, no theoretical or experimental work has been done for the effect of ocean environment on the local loss coefficient and local pressure drop. The simulation of local loss coefficient and local pressure drop in all kinds of components is of great importance in nuclear thermal hydraulic analysis. The local loss coefficient is also a necessary input parameter in nuclear engineering simulation.

In this paper, the local loss for a 90° bend in rolling motion is investigated with CFD code FLUENT. The calculation results are verified with experimental and theoretical results in steady state. The results of present study are hoped to shed some light on the local loss coefficient and local pressure drop in difficult components in ocean environment.

2. Theoretical models

2.1. Momentum equation

In rolling motion, the flow field and flowing behavior are influenced by the spatial and temporal variant additional forces.

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

0149-1970/$ – see front matter Crown Copyright © 2011 Published by Elsevier Ltd. All rights reserved.
doi:10.1016/j.pnucene.2011.11.006

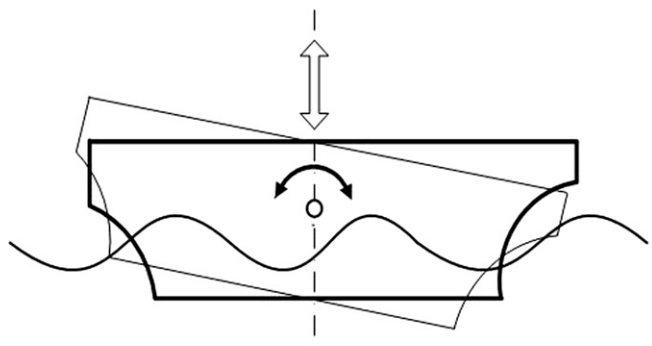

Fig. 1. Schematic of heaving and rolling motions.

The momentum equation of Reynolds time-averaged equations of turbulent flow should be modified as:

$$\frac{\partial}{\partial t}(\rho\overline{u_i})+\frac{\partial}{\partial x_j}\left(\rho\overline{u_i}\overline{u_j}\right)=-\frac{\partial\overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial\overline{u_i}}{\partial x_j}+\frac{\partial\overline{u_j}}{\partial x_i}\right)-\frac{2}{3}\delta_{ij}\frac{\partial\overline{u_j}}{\partial x_i}\right]+\frac{\partial}{\partial x_j}\left(-\rho\overline{u_i'u_j'}\right)+F_a \quad (1)$$

where the superscript $^-$ and $'$ denote the time-averaged and fluctuation values, respectively. u stands for velocity, t for time, ρ for density, p for pressure, x_k for coordinates. F_a is the additional force due to rolling motion, which can be expressed as:

$$F_a = \rho\vec{a} = \rho[\vec{\beta}\times\vec{r}+\vec{\omega}\times(\vec{\omega}\times\vec{r})+2\vec{\omega}\times\vec{u}] \quad (2)$$

where $\vec{\omega}$ and $\vec{\beta}$ stand for rolling angular velocity and acceleration, respectively. $\vec{r}$ is the vector radius. It is usually assumed that the rolling motion is in a sinusoidal order (Lewis, 1967). The rolling angular velocity and acceleration could be expressed as:

$$\omega = -\frac{2\pi\theta_m}{T}\sin\frac{2\pi t}{T} \quad (3)$$

$$\beta = -\frac{4\pi^2\theta_m}{T^2}\cos\frac{2\pi t}{T} \quad (4)$$

where θ_m and T are rolling amplitude and period, respectively.

2.2. Reynolds stress model

Abandoning the isotropic eddy-viscosity hypothesis, the Reynolds Stress Model (RSM) closes the Reynolds-averaged Navier–Stokes equations by solving transport equations for the Reynolds stresses, together with an equation for the dissipation rate (FLUENT Inc, 2005). Since the RSM accounts for the effects of streamline curvature, swirl, rotation, and rapid changes in strain rate in a more rigorous manner than one-equation and two-equation models, it has greater potential to give accurate predictions for complex flows. Adoption of RSM is also a must when the flow features of interest are the result of anisotropy in the Reynolds stresses. In rolling motion, the flow is affected by the spatial and temporal variant centrifugal force, tangential force and Coriolis force. The flow field is more complex than that in steady state. Therefore, the RSM is adopted in this work.

The turbulent kinetic energy is obtained by taking the trace of Reynolds stress tensor:

$$k = \frac{1}{2}\overline{u_i'u_i'} \quad (5)$$

An option is available in ANSYS FLUENT to solve a transport equation for the turbulence kinetic energy in order to obtain boundary conditions for the Reynolds stresses. In this case, the following model equation is used:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+\frac{1}{2}(P_{ii}+G_{ii})-\rho\varepsilon\left(1+2M_t^2\right)+S_k \quad (6)$$

where $\sigma_k = 0.82$ and S_k is a generic user-defined source term.

The dissipation tensor ε_{ij} is modeled as:

$$\varepsilon_{ij} = \frac{2}{3}\sigma_{ij}(\rho\varepsilon+Y_M) \quad (7)$$

where $Y_M = 2\rho\varepsilon M_t^2$ is the additional "dilatation dissipation" term. M_t is the turbulent Mach number.

The scalar dissipation rate ε is computed with a model transport equation which could be expressed as:

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{\varepsilon1}\frac{1}{2}[P_{ii}+C_{\varepsilon3}G_{ii}]\frac{\varepsilon}{k}-C_{\varepsilon2}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \quad (8)$$

where $\sigma_\varepsilon = 1.0$, $C_{\varepsilon1} = 1.44$, $C_{\varepsilon2} = 1.92$, $C_{\varepsilon3}$ is evaluated as a function of the local flow direction relative to the gravitational vector, and S_ε is a generic user-defined source term.

The turbulent viscosity μ_t is computed as:

$$\mu_t = \rho C_\mu k^2/\varepsilon \quad (9)$$

where $C_\mu = 0.09$.

2.3. Numerical scheme and boundary condition

The calculations presented in this work have been performed with the CFD code FLUENT 6.3. The numerical schemes adopted in the present case are:

1. the discretized equations are solved with a predictor corrector approach adopting the PISO algorithm.
2. temporal discretization is performed through an Implicit Euler scheme.
3. the third order Quadratic Upstream Interpolation of Convective Kinematics scheme (QUICK) has been adopted to guarantee the accuracy.
4. the low-Re stress-omega model is used to deal with the near wall boundary layer and the first near wall mesh has been kept at a value of $y^+ < 1.0$.

The sensitivity of calculation results on mesh structure is carefully checked and the nearly grid independent solutions are obtained. The velocity inlet and pressure outlet boundary conditions are applied. The no slip wall boundary condition with zero heat flux is adopted in this work. The Courant number has been kept below 0.3 to guarantee the accuracy.

3. Results and discussion

Energy losses occur when flow in a pipe is caused to change its direction. Consider the pipe bend illustrated in Fig. 2. Now whenever fluid flows in a curved path there must be a force acting radially inwards on the fluid to provide the inward acceleration. There is thus an increase of pressure near the outer wall of the