Annals of Nuclear Energy 78 (2015) 188–200

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Numerical analyses of flow distributions in nuclear fuel assemblies affected by grid deformations

Jong-Pil Park [1], Ji Hwan Jeong *

School of Mechanical Engineering, Pusan National University, Jangjeon-dong, Geumjeong-gu, Busan 609-735, Republic of Korea

ARTICLE INFO

Article history:
Received 17 September 2014
Received in revised form 30 December 2014
Accepted 2 January 2015
Available online 17 January 2015

Keywords:
Grid deformation
Subchannel blockage
Flow redistribution
Flow recovery
Computational fluid dynamics

ABSTRACT

In the event of a safety shutdown earthquake (SSE) in a nuclear power plant, the spacer grid of the fuel assembly will be deformed as a result of the vibrations. If the flow area in a subchannel is reduced due to the grid deformation, the coolant flow will be restricted and consequently a loss of flow occurs in the affected fuel assembly during the accident. In this study, computational fluid dynamics (CFD) analyses are conducted in order to assess the flow redistribution and flow recovery in fuel assemblies. The real geometries of an outer grid and mixing vane are used in the simulation, and the region including the inner grid is modeled as a porous media zone. The resistance coefficients of the porous media model are determined using CFD analyses. The Reynolds-averaged Navier–Stokes equation with a non-linear turbulence model was used to solve the three-dimensional anisotropic turbulence flow in the rod bundles during normal operation, blowdown, and reflood phases following a loss-of-coolant accident (LOCA). In these analyses, it is assumed that forty percent of the flow area is blocked by grid deformations. The results demonstrate that a downstream distance of 45 times the hydraulic diameter is required for the coolant flow to recover to 95% of the original flow rate in the affected fuel assembly.

1. Introduction

The coolant flow in a rod bundle is disrupted during subchannel blockages. The swelling or ballooning of the fuel rod cladding is of interest as a cause of subchannel blockages. If subchannel blockages occur, there will not be sufficient cooling water to cool the decay heat through the blocked subchannel because the deformed rod partially reduces the flow area in the subchannel and restricts the coolant flow during a loss-of-coolant accident (LOCA). Therefore, numerous studies have been conducted in order to understand the hydraulic characteristics in rod bundles with partial blockages due to swelling or ballooning of the fuel rod cladding (Creer et al., 1979; Ang et al., 1987, 1988a,b). These studies were conducted for adiabatic turbulent flows in 7×7 rod bundles containing deformed cluster arrays with various blockage ratios and blockage shapes. These results demonstrated that the coolant behavior in rod bundles is only dependent on the degree of blockage for high Reynolds numbers ($10{,}000 \leqslant Re \leqslant 58{,}000$). For low Reynolds numbers ($Re < 10{,}000$), however, the coolant flow is dependent on the blockage ratio as well as the Reynolds number. Despite these efforts, few studies have been conducted on the coolant flow in rod bundles that are affected by spacer grid deformations. Spacer grids are one of the most important structural components of fuel assemblies for thermal and mechanical performance enhancements, so they are designed to maintain their structural integrity under any design basis accidents (DBAs). If an unexpected serious safety shutdown earthquake (SSE) occurs, however, the spacer grids are readily deformed as a result of the nuclear reactor vibrations. Therefore, the coolant flow in the reactor core could be disrupted during the accident.

The robust fuel assemblies (RFAs) are used in the nuclear power plants (NPPs) at Ulchin, Korea. A new type of advanced fuel assembly called "ACE7" was developed, and it will replace the RFA fuels in the reactor power upgrade. Therefore, it is necessary to understand how the hydraulic behavior in the ACE7 fuel assembly is affected by spacer grid deformations in order to complete the safety evaluations of the nuclear fuel assembly and NPPs.

The recent progress in computer and computational fluid dynamics (CFD) technologies has enabled the CFD to be used for flow field analysis of real reactor geometry (Jeong and Han, 2008) and safety analyses of nuclear power plants (Park et al., 2011, 2012). Many numerical studies using three-dimensional (3D) CFD codes have been conducted in order to improve physical understandings of coolant flows in rod bundles. Lee and Choi (2007), Nematollahi and Nazihi (2008), Liu and Ferng (2010), and

* Corresponding author. Tel.: +82 51 510 3050; fax: +82 51 512 5236.
E-mail address: jihwan@pusan.ac.kr (J.H. Jeong).
[1] Current address: Korea Atomic Energy Research Institute, 1045 Daedeok-daero, Yusong, Daejeon 305-353, Republic of Korea.

http://dx.doi.org/10.1016/j.anucene.2015.01.004
0306-4549/

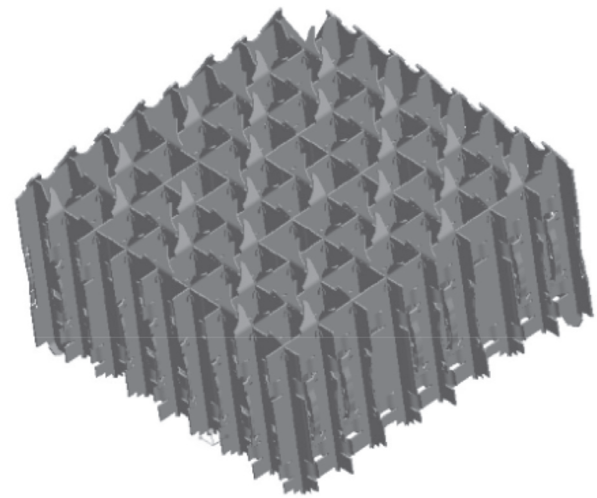
Liu et al. (2012) have conducted CFD analyses for turbulence flows around mixing vanes. Tzanos (2002), Roelofs et al. (2012), Salama (2012), and Salama and El-Morshedy (2012) have conducted CFD analyses to predict the coolant behavior in various types of fuels, such as typical pressurized water reactor (PWR) fuels and plate-type fuels in research reactors containing partial blockages. The CFD codes have also been widely used in other engineering disciplines as well as nuclear safety analyses. Han et al. (2014) used a commercial CFD package to investigate the flow distribution of refrigerants inside a two-phase refrigerant distributor. The CFD analysis results appear to be in good agreement with the experimental data, even though these results are dependent on turbulence models.

This study evaluates the coolant behavior in ACE7 fuel assemblies, particularly the distance in which the coolant flow rate is recovered to the original flow rate. It was assumed that 40% of the flow area of an affected ACE7 fuel assembly is blocked due to a MID grid deformation while the surrounding assemblies are intact. The CFD analyses were performed in three different flow conditions: normal operating conditions, blowdown phase, and reflood phase following a LOCA. The region including the inner grids is represented as porous media in order to reduce the computing resources required. In contrast, the outer grid and mixing vanes are modeled to have real geometries because these parts significantly influence the swirling and turbulence of coolant flows. Thus, this paper consists of two parts. The first part is the unit subchannel analyses that evaluate the resistance coefficients for the inner grid region represented by the porous media. The second part is the overall CFD analyses of coolant flows in the ACE7 fuels affected by spacer grid deformations in order to evaluate the hydraulic characteristics such as flow redistribution and recovery.

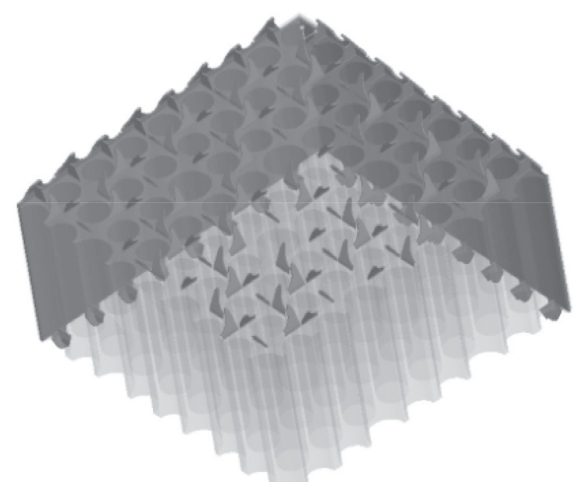
2. Numerical methodology

Because spacer grids have very complex features, performing CFD analyses for coolant flows in full-scale PWR fuel assemblies using real geometries requires significant computing resources. For example, Kwack et al. (2013) used approximately 0.2 billion computational meshes for the CFD analysis of a coolant flowing through a full length 16×16 fuel assembly without modeling the springs and dimples of the spacer grid in low flow conditions. It required more than 40 days to obtain a converged solution, even with using a parallel computer with 138 cores. Therefore, it appears to be more practical to analyze coolant flows in a fuel assembly using a porous media approach. In this study, the region including the inner grid of the ACE7 fuel is represented using a numerical model of the porous media in order to reduce the computing burden, while the fuel rods and mixing vanes are represented using the same geometry as real fuel assemblies. Fig. 1 presents a schematic that identifies the numerical porous media model and the real geometry. Because the porous media model is a numerical model, it is necessary to confirm whether it is well designed for the present analysis.

The CFD analyses for the present work proceed in three steps: validation of CFD models, CFD analysis for a subchannel (unit subchannel analysis), and overall CFD analysis for the coolant flowing through the ACE7 fuel assembly (i.e. flow blockage analysis). In the first step, the numerical models were validated through comparing the CFD analysis results against the experimental measurements in terms of pressure loss coefficients. In the second step, CFD analyses were performed using the same numerical methods and models as those determined in the first step in order to obtain the parameters associated with the numerical porous media models. Lastly, in the third step, the numerical porous media models were used to represent the grids of the ACE7 fuel assembly and coolant flow fields in the fuel assembly are analyzed. The purpose of the flow blockage analysis is to evaluate the coolant flow behaviors such as flow redistribution and recovery in ACE7 fuel assemblies with deformed MID grids.



(a) 1/4 CAD model for the MID grid.



(b) Region represented as a porous media model.

Fig. 1. Representation of the MID grid geometry for the CFD analyses.

3. Numerical model setup

3.1. Unit subchannel analysis

3.1.1. Numerical model

A typical coolant centered subchannel surrounded by four fuel rods is constructed as a unit subchannel model. The geometry of this unit subchannel model has the same geometry as the ACE7 fuel assembly including the fuel support grids and mixing vanes. Fig. 2 illustrates the schematics of the computational domains of the unit subchannel for the MID grid and IFM grid. The entrance region and downstream region are introduced and are sufficiently long to provide a fully developed flow at the inlet and outlet in order to obtain an appropriate converged solution. Furthermore,