Annals of Nuclear Energy 112 (2018) 71–83

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Analysis of flow blockage accidents in rectangular fuel assembly based on CFD methodology

Zhenhui Ma [a], Ronghua Chen [a], Maolin Tian [a], Jian Song [a], Dalin Zhang [a], G.H. Su [a,*], Xiaorong Li [b], Linglan Zhou [b], Suizheng Qiu [a]

[a] School of Nuclear Science and Technology, State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China
[b] Nuclear Power Institute of China, Chengdu 610213, China

ARTICLE INFO

Article history:
Received 25 May 2016
Received in revised form 2 September 2017
Accepted 8 September 2017

Keywords:
CFD
Rectangular fuel assembly
Flow blockage
DNB

ABSTRACT

Flow blockage accidents in rectangular fuel assembly were investigated by three-dimensional CFD method in detail. Totally six coolant channels of the fuel assembly were modeled. Meanwhile, in order to make understand of the thermal hydraulic characteristics of the flow blockage accidents comprehensively, three coolant channels which included the obstructed channel were investigated and analyzed thoroughly based on simulation results. RELAP5 calculation was performed here to compare with CFD simulation results under non-blocked condition, and the comparison results indicated approximate agreement of such two types of results. On the basis of the CFD simulation, velocity and temperature profiles were discussed for some typical blockage cases, and conclusion was drawn that the redistribution of the mass flow rates occurred after the formation of the blockage, and due to the formation of obstruction, temperature of the coolant and the fuel increased rapidly which caused higher peak temperature in the blockage channel. Simultaneously, the increasing flow resistance would lead to the existence of jet-flow and reverse flow in the obstructed channel. In addition, DNBR calculation indicated that heat flux on the cladding surface would not exceed critical heat flux, thus DNB would not occur under the investigated situation.

1. Introduction

Nowadays the rectangular fuel assembly is widely used in research reactor and integrated reactor in that it owns compact structure, higher thermal efficiency and lower temperature in the fuel assembly. In the rector which uses rectangular fuel assemblies, water flows along several parallel coolant channels and heat released by the fuel assembly can be transferred outside the reactor core in time, which is similar to the typical pressurized water reactor.

However, after using such reactors for a long time, the mechanical impurities and other debris may be easily taken into the coolant channel, which may cause flow blockage. The flow blockage accidents can also be incurred by other potential cases, such as buckling of the fuel plates due to thermal stress. Unlike the typical pressurized water reactor, the flow blockage in the obstructed channel will not lead to apparent decrease in the total mass flow in reactor core (Wenyuan et al., 2015). As a result, this kind of accident only causes loss of flow accident (LOFA) which can lead to rise of the peak temperature of the fuel-plates and redistribution of the coolant in parallel coolant channels. Thus the outlet temperature of the reactor core and the total mass flow rate will not change dramatically, which makes it more difficult to troubleshoot the flow blockage accident. Therefore, the investigation into the aftermath of the flow blockage is extremely crucial for design and safe operation of such reactors because of its critical role in the safety analysis targeting the rectangular fuel assembly.

Experiments were carried out to study flow blockage accidents in different kinds of nuclear reactors (Ohtsubo and Uruwashi, 1972; Kikuchi et al., 1977; Sudo and Osakabe, 1983). However, none of these experiment could provide detailed information about coolant velocity and temperature distribution due to the limitation of measurement technology. The structure of the rectangular fuel assembly is compact and the width of the coolant channel is in millimeter level, so it is impossible to carry out experiments of this kind to obtain effective experimental data. This explains why the numerical simulation method was widely adopted in this research field.

* Corresponding author.
E-mail address: ghsu@mail.xjtu.edu.cn (G.H. Su).

https://doi.org/10.1016/j.anucene.2017.09.012
0306-4549/

Nomenclature

A_b	blockage area of the flow channel
A_t	total area of the flow channel
br	blockage ratio
$DNBR$	Departure from Nucleate Boiling Ratio
D_ω	cross-diffusion term
G_k	production of k
G_ω	production of ω
k	turbulent kinetic energy
$MDNBR$	minimum value of DNBR
q	current heat flux
q_{DNB}	critical heat flux
S_k	source term of k
S_ω	source term of ω
u	fluid velocity
x	coordinate position
Y_k	dissipation of k due to turbulence
Y_ω	gissipation of ω due to turbulence

Greek symbols

ρ	density
$\delta_{i,j}$	Kronecker delta
ω	dissipation rate
υ_t	eddy viscosity
Γ_k	effective diffusivity of k
Γ_ω	effective diffusivity of ω

Subscripts

i, j	direction of component

There are two different types of numerical simulation methods which are all representative for the engineering application. The first method, the utilization of one-dimensional codes is broadly used in nuclear engineering analysis, such as the application of RELAP5 in reactor safety analysis by appling massive empirical correlations to analyze these codes. Adorni's (Adorni et al., 2005) calculation indicated that blockage ratio would influence the physical state of the coolant in the obstructed channel, therefore, high blockage ratio would lead to local dry out which may pose threats for the integrity of the fuel. However, due to the imperfection of the physical model, the calculation results were conservative to some extent. Hyung et al. (2015) found that the oscillation of the fuel temperature was observed in some cases due to the cycling variation of vapor inventory inside the obstructed channel. Qing et al. (2009) developed a thermal hydraulic analysis code, THAC-PRR, to analyze the 95% blockage accident. According to the research finding, the reduction of the mass flow rate in the obstructed channel, which may alleviate the consequences of flow blockage, would intensify the heat conduction along the normal direction of coolant flow. Khan et al. (2014) used neutron kinetics and thermal hydraulic coupling code to analyze flow blockage accidents, and verified the simulation results with RELAP5, and results procured from both techniques are nearly the same. Although the application of analysis codes is relatively developed, the deficiency of this method is evident that it is not possible to simulate the sophisticated local phenomenon in the coolant channel. Besides using RELAP5, the MELCOR, an effective code for severe accident exploration, was also used in flow blockage analysis inside the reactor core. Wang et al. (2016) studied the degradation in the reactor core with the aid of MELCOR and it was found that flow channels in the bottom part of the core were always completely blocked, which prevented the severe accident mitigation.

The second method is the application of CFD (computational fluid dynamics) technology. The CFD method is specializing in simulating the local phenomenon in the coolant channel. Thus, the research field of simulation study is promising. Amgad Salama made a great effort in the study of this field through CFD simulation (Salama, 2011, 2012; Salama and El-Morshedy, 2011, 2012a, b; Salama et al., 2015) and reported 2D and 3D results towards the aftermath of flow blockage accidents. According to the research, when the blockage ratio in the obstructed channel exceeded 80%, coolant boiling may occur and the integrity of the fuel would be influenced. Gong et al. (2015) mainly researched heat transfer characteristics of high flux test reactor and found that fuel and cladding temperature are both below the design limits, which can prove the assembly in a safe state. Davari et al. (2015) also studied the buckling of the plate-type fuel by means of the CFD method. They found that critical phenomenon would occur and the clad integrity would be compromised simultaneously when cladding temperature in the obstructed channel exceeded 105 °C. Besides, nucleate boiling could be observed in the obstructed channel when the blockage ratio reached 70%, which would affect the reactor safety. By means of the CFD method, Yan et al. (2011) mainly investigated the flowing characteristics of the turbulent flow in the rectangular fuel assembly in ocean environment. In the simulation process, the additional forces caused by the ocean environment were taken into consideration, which resulted in more averaged velocity profile compared with land-based nuclear power systems. Wenyuan et al. (2015) conducted the transient simulation with the aid of CFD method and found that transient process would happen immediately after the formation of the blockage. Meanwhile, jet-flow and vortex could be observed in the vicinity of the blockage. Wenyuan et al. (2016) wrote in their research paper that they employed the CFD method to thoroughly investigate inlet flow blockage accidents in the plate-type fuel assembly, and applied the porous-jump treatment condition to the simulation for the first time. In addition, RELAP5 calculations were also conducted in the simulation.

Although many studies were carried out to research blockage accidents in the rectangular channel, a vast majority of the simulation work just focused on the buckling of the fuel. Furthermore, researchers pay more attention to the flow blockage accident in the inlet of the blocked channel, which is extremely incomprehensive.

Accordingly, most studies done by only a few researchers about the aftermath of such accidents through three-dimensional CFD method are not that thorough enough. The present study took up the 3D simulation way for a comprehensive study of the blockage accidents caused by the debris, and the blockage at different heights of the blocked channel. In addition, the study also investigated the flow blockage accidents under different power distributions and got some results that were of much significance. More realistic and accurate results obtained through CFD simulation means the necessity to apply this promising method.

2. Statement of the simulation

2.1. Description of the fuel assembly

The single fuel assembly is composed of five parallel plate-type fuels, four of which constitute the central coolant channels. Two