Annals of Nuclear Energy 114 (2018) 413–426

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Studies on the subcooled boiling in a fuel assembly with 5 by 5 rods using an improved wall boiling model

Tenglong Cong [a,*], Rui Zhang [b], Lijuan Chen [a], Xiang Zhang [a], Ting Yu [a]

[a] Fundamental Science on Nuclear Safety and Simulation Technology Laboratory, Harbin Engineering University, 145 Nantong St, Harbin 150001, China
[b] Department of Engineering Physics, University of Wisconsin-Madison, Madison, WI 53706, USA

ARTICLE INFO

Article history:
Received 21 June 2017
Received in revised form 12 December 2017
Accepted 27 December 2017

Keywords:
Flow boiling
Wall boiling model
Fuel assembly
Critical heat flux

ABSTRACT

An improved wall boiling model with consideration of thin film heat transfer was employed with the Eulerian two-fluid model to investigate the three-dimensional flow boiling characteristics in a fuel assembly with 5 by 5 rod bundle and a vaned grid. Models were validated by using the experimental data for subcooled boiling, post-dryout heat transfer and critical heat flux. The calculated results agreed well with experimental data. Thermohydraulics in the rod bundle were obtained, including temperature, velocity, phasic volume fraction and pressure, based on which, the effects of mixing vane on the localized thermohydraulics were studied. Mixing vane can significantly reduce the cross section averaged vapor fraction and increase the heat transfer capacity due to the swirl effects on two-phase flow; however, it will increase the localized vapor fraction on the heated surface, which may result in the anticipation of boiling crisis, i.e., reach the critical heat flux. Besides, influences of heat flux profile on the flow and heat transfer characteristics were also obtained by comparing two cases with uniform and cosine heat flux distribution along the axial direction. Finally, the impacts of thin film heat transfer on the wall heat partition were investigated, proving that thin film heat transfer played an essential role in the modeling of wall boiling when the vapor fraction at heated surface exceeding 0.25.

1. Introduction

In a nuclear power plant, the fuel assembly in the core will generate heat by atomic fission of the fissile nuclide. Meanwhile, the fission reaction will produce radioactive substances, most of which are contained in the reactor pin inside the fuel cladding. These radioactive materials may be released to the primary loop, the containment or the atmosphere under certain conditions such as cladding rupture and core degradation. The integrity of the fuel assembly should be kept during the steady-state operation, normal transient operation, design basis accident (DBA) and even severe accident (SA) conditions to prevent the radioactivities release. However, fuel assembly encounters adverse working conditions during operation, such as the flow-induced vibration, neutron exposure, fretting, chemical corrosion, burn out, and even meltdown. The corresponding experiments and analysis should be carried out to ensure the safety of nuclear fuel assembly prior to licensing or constructing. Among these, the thermohydraulics analysis in the fuel assembly is one of the foundations of other analyses. For instance, the chemical corrosion analysis can only be performed based on the localized temperature, velocity and oxygen content in the water, which are obtained from the thermohydraulics analysis.

In the past few decades, numerous work had been published on the thermohydraulics in the fuel assembly by using experimental, theoretical and numerical methodologies. Ikeda and Hoshi (2007) measured the flow parameters in a fuel assembly with spacer grids to investigate the cross flow induced by spacer grids in a rod bundle, and found that the grid straps and the mixing vane control the localized flow distribution near the spacer grids. Navarro and Santos (2011) carried out a CFD study by CFX code to investigate the influences of spacer grid on the localized heat transfer performance and to evaluate the numerical procedure for the CFD simulation on flow in the fuel assembly. However, most of these work focused on the single-phase flow since boiling flow was not allowed under the normal operation condition in the PWR core for the generation one or two power plants.

With the development of the nuclear power plants, subcooled boiling is allowed in the hot channel of the core in the up-to-date reactor design, such as the AP1000 power plant (Winters et al., 2004). Subcooled boiling refers to the condition where the wall superheat is large enough to generate and enlarge the vapor bubbles on heated surface while the main stream temperature is

* Corresponding author.
E-mail address: tlcong@hrbeu.edu.cn (T. Cong).

https://doi.org/10.1016/j.anucene.2017.12.058
0306-4549/

Nomenclature

A_b	area fraction of heated wall influenced by the bubbles
c_p	specific heat at constant pressure
D_h	hydraulic diameter
f	bubble departure frequency
F_D	drag force
$F_{L,i}$	lift force
F_{td}	turbulence dispersion force
F_{vm}	virtual mass force
F_{wl}	wall lubrication force
g	gravity
h	enthalpy/heat transfer coefficient
h_{fg}	latent of vaporization
k_l	liquid conductivity
m	mass
$\vec{n}_w$	unit normal pointing way from wall
p	pressure
q	heat flux
q_c	liquid convective heat flux
q_e	the evaporate heat flux
q_g	vapor convective heat flux
q_q	quenching heat flux
q_f	thin film heat flux
S	source term
T	temperature
T^+	non-dimensional temperature
t	time
v	velocity
Y^+	non-dimensional wall distance

Greek symbols

α	volume fraction
δ_f	thickness of thin film
μ	viscosity
ρ	density

Subscripts

g	gas
i	liquid or gas Phase
l	liquid
sat	saturation
sub	subcooled fluid
w	wall

Abbreviations

CHF	Critical Heat Flux
DBA	Design Basis Accident
Pr	Prandtl number
Re	Reynolds number

still lower than the saturated temperature. Subcooled and saturated boiling flow can enhance the heat transfer capacity by orders of magnitude compared with the single-phase convective flow. Thus, boiling flow is widely used in the industrial applications, including in the newest PWR core. However, the heat transfer capability enhancement by evaporation heat transfer is restricted by the boiling crisis, which might be caused by the bubble aggregation on the heated surface. The heat flux that can trigger the boiling crisis is named critical heat flux (CHF), beyond which the heat transfer capacity will be deteriorated and the wall temperature may increase by hundreds degrees. Fuel cladding may be burnt up under this condition, which may result in the hydrogen generation or the radioactivity release. Besides, the density of vapor phase generated by boiling is much less than the liquid. The mixture comprised by two fluids with large density difference will amplify the flow induced vibration and may lead to the rupture of fuel cladding. Therefore, the boiling characteristics, including the localized velocity, phasic volume fraction and CHF should be investigated throughout for the design and operation of reactor.

Similar to the single phase flow, two-phase flow characteristics can be investigated by using experimental, theoretical and numerical methods. The experiments on the two-phase boiling flow in fuel rod bundles can present the characteristics in reactor core in a most realistic way. Researchers can measure the rod temperature, fluid temperature, pressure and velocity or mass flux, calculate the pressure drop coefficient and heat transfer coefficient and evaluate the design scheme. However, it is time and money consuming. Besides, it is difficult to measure the localized two-phase parameters among the rod bundle and spacer grids under the conditions with high temperature and high pressure, which are essential for the design and safety analysis. The empirical correlations can be derived from the experimental data and theoretical analysis, and be used in the safety analysis. Nevertheless, the applicable range of these experiment-based correlations is strictly limited by the experimental conditions, i.e., these correlations can only be used under the conditions which are covered by the experiments, rather than new designs or new working conditions. As for the theoretical analysis, it is used to be applied to the extremely simple flow channels, such as the circular channel. It is unpractical to use a theoretical analysis to predict the localized thermohydraulics analysis in the fuel assembly with spacer grids.

Various numerical methodologies are employed in the design and safety analysis of nuclear power plants related to the two-phase flow in a PWR or BWR fuel assembly, including the system analysis, subchannel analysis and CFD analysis. The system analysis focuses on the systematical responses of the nuclear power plant under normal transients, DBAs, beyond DBAs and SAs, while the localized thermohydraulics are not the priorities. The subchannel code can predict the two-phase flow and heat transfer parameters within each channel with good accuracy. However, the subchannel code models the spacer grid by using coefficients obtained from corresponding experiments, such as cross flow coefficients, which are geometry sensitive and cannot be extended to new designs without further experiment validations.

CFD technology is considered to be a universal tool for the design and analysis of fuel assembly for single-phase flow since it doesn't use coefficients depending on geometry and work conditions. High-fidelity CFD code with sufficient validation can be used to carry out "numerical experiments", which is a promising tool to substitute the experiments under certain conditions. Chu et al. (2016) employed a direct numerical simulation to investigate the air flow in a vertical tube with large wall heat flux under the condition of moderate Reynolds number. When it comes to the two-phase flow conditions, CFD technology is also one of the most promising methodologies can be used without the limitation from geometry and working conditions (Bestion, 2010). However, the investigation on two-phase CFD technology is more immature when compared with that on single-phase problems.

Two methodologies can be employed to simulate the two-phase flow, that is, the interface tracking method and the space-averaged method. The interface tracking method can recognize the interface between two-phase and track the time-varying interface, including the front tracking method (Popinet and Zaleski, 1999), volume of fraction method (Bartosiewicz et al., 2008), level-set method (Wu