Technical Note

Numerical investigation of the critical heat flux in a $5 \times 5$ rod bundle with multi-grid

Wei Liu a, Zemin Shang b, Shihao Yang b, Lixin Yang b,*, Zihao Tian b, Yu Liu a, **, Xi Chen a, Qian Peng a

a State Key Laboratory of Reactor System Design Technology, Nuclear Power Institute of China, Chengdu, 610213, China
b School of Mechanical, Electronic and Control Engineering, Beijing Jiaotong University, Beijing, 100044, China

A R T I C L E   I N F O

Article history:
Received 23 March 2021
Received in revised form 7 October 2021
Accepted 15 November 2021
Available online 17 November 2021

Keywords:
Eulerian two-fluid model
Critical heat flux
$5 \times 5$ rod bundle
Flow boiling

A B S T R A C T

To improve the heat transfer efficiency of the reactor fuel assembly, it is necessary to accurately calculate the two-phase flow boiling characteristics and the critical heat flux (CHF) in the fuel assembly. In this paper, a Eulerian two-fluid model combined with the extended wall boiling model was used to numerically simulate the $5 \times 5$ fuel rod bundle with spacer grids (four sets of mixing vane grids and four sets of simple support grids without mixing vanes). We calculated and analyzed 31 experimental conditions under different pressure, inlet temperature, and mass flux. After comparing the CHF and the location of departure from the nucleate boiling obtained by the numerical simulation with the experimental results, we confirmed the reliability of computational fluid dynamic analysis for the prediction of the CHF of the rod bundle and the boiling characteristics of the two-phase flow. Subsequently, we analyzed the influence of the spacer grid and mixing vanes on the void fraction, liquid temperature, and secondary flow distribution. The research in this article provides theoretical support for the design of fuel assemblies.

© 2021 Korean Nuclear Society, Published by Elsevier Korea LLC. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

1. Introduction

To improve the heat transfer efficiency of the reactor fuel assembly, phase change has been allowed in the design of advanced reactors. The outlet of the hot channel in the AP1000 reactor can be in a state of saturation boiling under normal operation condition [1]. With an increase in local heat flux on the fuel rod surface, the boiling crisis named “departure from nucleate boiling” (DNB) under low-quality regions will cause the deterioration of heat transfer. This effect rapidly decreases the efficiency of the heat transfer, and the rod surface temperature may increase rapidly by hundreds of degrees. The corresponding heat flux is called the critical heat flux (CHF). Accurate prediction of CHF in a reactor is one of the key issues affecting reactor fuel assembly design and operational safety.

In the past few decades, the experimental research on the phenomenon of flow boiling and DNB has been carried out mainly in simple geometric structures (such as round tubes, annular tubes, and rectangular channels), and empirical or semi-empirical correlations have been developed on the basis of experiments. For water-cooled tubes alone, more than 1000 empirical formulas have been developed, which shows the complexity of the CHF mechanism [2]. When designing fuel assemblies, however, it is difficult, costly, and time-consuming to conduct a CHF experiment in a rod bundle under high temperature and high pressure conditions. Moreover, the empirical relationship obtained from the experiment has limitations and can be applied only to specific geometric structures and working conditions.

With the continuous development of computational fluid dynamics (CFD) and computer science, scholars have developed an increasing number of numerical models for simulation calculations under complex geometries and working conditions. The substantial improvement of computer hardware and computing capacity also has provided support for the calculation of real flow characteristics in the fuel assembly. Table 1 lists the relevant research on the numerical simulation of flow characteristics and thermal-hydraulic performance in fuel assemblies based on the CFD method in recent years. In these studies, Some studies used the CFD analysis tools to conduct a numerical simulation on the single-phase flow heat transfer of the fuel rod bundle containing the spacer grids.

https://doi.org/10.1016/j.net.2021.11.018
1738-5733/© 2021 Korean Nuclear Society, Published by Elsevier Korea LLC. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).
<table>
<thead>
<tr>
<th>Author(s)</th>
<th>Geometry</th>
<th>Fluid</th>
<th>Operating conditions</th>
<th>Turbulent model</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kazuo Ikeda [1]</td>
<td>5 x 5 rod bundle</td>
<td>Water</td>
<td>P = 10.3 MPa</td>
<td>Standard k−ε</td>
<td>Single-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 0.508 m</td>
<td></td>
<td>Inlet Velocity = 4.83 m/s</td>
<td>Complex geometry</td>
<td>STAR-CD</td>
</tr>
<tr>
<td>Moysés A. Navarro [4]</td>
<td>5 x 5 rod bundle</td>
<td>Water</td>
<td>P = 0.483 MPa</td>
<td>k−ε model</td>
<td>Single-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 0.66 m</td>
<td></td>
<td>Inlet Velocity = 6.79 m/s</td>
<td>Complex geometry</td>
<td>CFX 11.0</td>
</tr>
<tr>
<td>C.C. Liu [5]</td>
<td>5 x 5 rod bundle</td>
<td>Water</td>
<td>P = 0.101 MPa</td>
<td>RNG k−ε</td>
<td>Single-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 0.1016 m</td>
<td></td>
<td>Inlet Velocity = 2.4−3.6 m/s</td>
<td>Realizable k−ε</td>
<td>Fluent 12.0</td>
</tr>
<tr>
<td>Deqi Chen [6]</td>
<td>5 x 5 rod bundle</td>
<td>Water</td>
<td>P = 7.4 MPa</td>
<td>Standard k−ε</td>
<td>Single-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 0.4932 m</td>
<td></td>
<td>Inlet Velocity = 2.78 m/s</td>
<td>Realizable k−ε</td>
<td>Complex geometry</td>
</tr>
<tr>
<td>Yingjie Wang [7]</td>
<td>5 x 5 rod bundle</td>
<td>Water</td>
<td>P = 15.52 MPa</td>
<td>Standard k−ε</td>
<td>Single-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 0.508 m</td>
<td></td>
<td>Inlet Velocity = 1.5 m/s</td>
<td>Realizable k−ε</td>
<td>Complex geometry</td>
</tr>
<tr>
<td>Songbai Cheng [8]</td>
<td>5 x 5 rod bundle</td>
<td>Water</td>
<td>P = 15 MPa</td>
<td>SST k−ω</td>
<td>Single-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 1.12 m</td>
<td></td>
<td>Inlet Velocity = 1 m/s</td>
<td>Complex geometry</td>
<td>FLUENT</td>
</tr>
<tr>
<td>Jin Lei [9]</td>
<td>7 x 7 rod bundle</td>
<td>Water</td>
<td>P = 14.5 MPa</td>
<td>RNG k−ε</td>
<td>Single-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 0.59 m</td>
<td></td>
<td>Inlet Velocity = 4.13 m/s</td>
<td>Scalable wall function</td>
<td>Complex geometry</td>
</tr>
<tr>
<td>Rui Zhang [10]</td>
<td>Vertical tube</td>
<td>Water</td>
<td>P = 4.5 MPa</td>
<td>k−ε model</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 2 m</td>
<td></td>
<td>G = 900 kg/(m² s)</td>
<td>k−ω model</td>
<td>Simple geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>Fluent 14.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>L = 2 m</td>
<td></td>
<td>G = 994 kg/(m² s)</td>
<td>Simple geometry</td>
<td>Star-CCM</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>Fluent 12.1</td>
<td></td>
</tr>
<tr>
<td>J.S. Muralidhara [12]</td>
<td>Vertical tube</td>
<td>Water</td>
<td>P = 4.5 MPa</td>
<td>Standard k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 2 m</td>
<td></td>
<td>G = 994 kg/(m² s)</td>
<td>Simple geometry</td>
<td>Star-CCM</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>Fluent 14.5</td>
<td></td>
</tr>
<tr>
<td>R. Thakrar [13]</td>
<td>Rectangular channel</td>
<td>Water</td>
<td>P = 4.1 MPa</td>
<td>Standard k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 1.26 m</td>
<td></td>
<td>G = 900 kg/(m² s)</td>
<td>Reynolds stress</td>
<td>Simple geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>Transport</td>
<td>Star-CCM</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Fluent 17.2</td>
<td></td>
</tr>
<tr>
<td>Chaitanya R. Mali [14]</td>
<td>Vertical tube</td>
<td>Water</td>
<td>P = 1.46−15 MPa</td>
<td>Standard k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 2−13.4 m</td>
<td>R12</td>
<td></td>
<td>Realizable k−ε</td>
<td>Simple geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>Fluent 14.5</td>
<td></td>
</tr>
<tr>
<td>Quan Li [15]</td>
<td>Vertical tube</td>
<td>Water</td>
<td>P = 1.5−15 MPa</td>
<td>SST k−ω</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 2 m</td>
<td></td>
<td>G = 300−1000 kg/(m² s)</td>
<td>Complex geometry</td>
<td>FLUENT 14.5</td>
</tr>
<tr>
<td>Rui Zhang [16]</td>
<td>Vertical tube</td>
<td>Water</td>
<td>P = 0.1−2.5 MPa</td>
<td>Realizable k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 0.1 m</td>
<td></td>
<td>Inlet Velocity = 10−40 m/s</td>
<td>Enhanced wall function</td>
<td>Simple geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>Fluent 14.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>L = 0.0475 m</td>
<td></td>
<td>G = 940−2650 kg/(m² s)</td>
<td>Simple geometry</td>
<td>Star-CCM</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>Fluent 14.5</td>
<td></td>
</tr>
<tr>
<td>Xiaomeng Dong [18]</td>
<td>2 x 2 rod bundle</td>
<td>Water</td>
<td>P = 13.8 MPa</td>
<td>Realizable k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 2.17 m</td>
<td></td>
<td>G = 940−2650 kg/(m² s)</td>
<td>Simple geometry</td>
<td>Star-CCM</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>Fluent 16.0</td>
<td></td>
</tr>
<tr>
<td>Xiaomeng Dong [19]</td>
<td>Vertical tube</td>
<td>Water</td>
<td>P = 7.01 MPa</td>
<td>RNG k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 7 m</td>
<td></td>
<td>G = 1495.3 kg/(m² s)</td>
<td>Realizable k−ε</td>
<td>Simple geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>Fluent 14.5</td>
<td></td>
</tr>
<tr>
<td>Quan Li [20]</td>
<td>Vertical tube</td>
<td>Water</td>
<td>P = 12−16 MPa</td>
<td>Standard k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 1−1.83 m</td>
<td></td>
<td>G = 2000−4000 kg/(m² s)</td>
<td>All y+ wall function</td>
<td>Simple geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform and nonuniform heat flux</td>
<td>Star-CCM</td>
<td>FLUENT 16.1</td>
</tr>
<tr>
<td>Harish Pothukuchi [21]</td>
<td>Vertical tube</td>
<td>Water</td>
<td>P = 7 MPa</td>
<td>RNG k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>Annulus</td>
<td></td>
<td>G = 500−2000 kg/(m² s)</td>
<td>Standard k−ε</td>
<td>Simple geometry</td>
</tr>
<tr>
<td></td>
<td>4 × 4 rod bundle</td>
<td></td>
<td>Uniform and nonuniform heat flux</td>
<td>SST k−ω</td>
<td>FLUENT 14.5</td>
</tr>
<tr>
<td></td>
<td>L = 3.65−7 m</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>S.R.G. Vadlamudi [22]</td>
<td>Vertical tube</td>
<td>Water</td>
<td>P = 13.8 MPa</td>
<td>k−ε model</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>Annulus</td>
<td></td>
<td>G = 1351−2715 kg/(m² s)</td>
<td>Enhanced wall function</td>
<td>Simple geometry</td>
</tr>
<tr>
<td></td>
<td>4 × 4 rod bundle</td>
<td></td>
<td>Uniform heat flux</td>
<td>Fluent 16.1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>L = 0.4572 m</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Byung Soo Shin [23]</td>
<td>2 x 3 rod bundle</td>
<td>Water</td>
<td>P = 1.5−2.5 MPa</td>
<td>Standard k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td>L = 1.125 m</td>
<td></td>
<td>G = 300−400 kg/(m² s)</td>
<td>Complex geometry</td>
<td>Complex geometry</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td>CFX 11.0</td>
<td></td>
</tr>
<tr>
<td>Seung Jun Kim [24]</td>
<td>5 x 5 rod bundle</td>
<td>Water</td>
<td>P = 14.5 MPa</td>
<td>Standard k−ε</td>
<td>Two-phase flow</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>G = 900−1000 kg/(m² s)</td>
<td>Complex geometry</td>
<td>FLUENT 14.5</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Uniform heat flux</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
[3–9]. The tested assembly included three scales (i.e., 1 × 3, 5 × 5, and 7 × 7 rod bundle) but considered only one spacer grid. In the research of flow heat transfer characteristics in complex structures, the flow field, temperature distribution, pressure drop, and thermal hydraulic performance obtained by the CFD calculation have been compared with experimental results to verify the reliability of CFD analysis tools. Valuable suggestions have been given based on the calculation results, for example, the $k$-$\varepsilon$ turbulence model combined with the enhanced wall treatment method can obtain more accurate results and better robustness. The complex geometry of the spacer grid region was meshed with unstructured meshes to capture the local flow information, whereas in the upstream and downstream areas of the spacer grid, the use of structured meshes could reduce the number of grid cells, thereby increasing the calculation speed and reducing the calculation overhead.

Some researchers numerically simulated the subcooled flow boiling in simple structures such as vertical tubes or rectangular channels using CFD analysis tools based on the Eulerian two-fluid model [10–15]. The researchers conducted a comprehensive and careful comparison of turbulence models, phase interaction models, and wall heat flux partitioning models within the framework of the Eulerian two-fluid model. The numerical simulation under different pressure, mass flux, and heat flux distribution were conducted, which fully demonstrated the applicability of the Eulerian two-fluid model under subcooled flow boiling conditions in simple structures.

The researchers’ confidence in their results was encouraged by the accurate calculation of the phenomenon of subcooled flow boiling in simple geometric structures. Subsequently, the wall boiling model in the Eulerian two-fluid framework was improved to take into account the single vapor phase convection heat flux so that the improved wall boiling model could be used for DNB predictions. Table 1 summarized some CHF prediction research in simple structures based on the improved wall boiling model [16–22]. According to the research results, the Eulerian two-fluid model has been able to predict the DNB phenomenon in a simple structure with a wide range of working conditions (pressure range: 0.1–16 MPa, mass flux range: 500–4000 kg/m²s).

Some scholars attempted to apply the Eulerian two-fluid model to the calculation of flow boiling and CHF in complex structures (such as a bundle channel with mixing vanes) [23–26]. According to the literature summarized in this article, most of these studies were case by case calculations and only confirmatory calculations were carried out for a single working condition. Only R. Zhang et al. calculated seven working conditions under different mass flux and inlet subcooling [1]. The maximum deviation of the results was approximately 26%. The Eulerian two-fluid model for CHF prediction in complex structures such as fuel assembly with wide applicability and high calculation accuracy, requires further study.

Through this analysis, we found that the current prediction of CHF in the simple channel has achieved more results, and the predicted CHF was in good agreement with the experimental. The current research on two-phase flow in complex geometric structures, however, remains insufficient, and related research on the prediction of the CHF in complex structures such as fuel assembly is even more scarce. In this study, we used the commercial CFD package STAR-CCM+ to simulate the two-phase flow and predict CHF in a $5 \times 5$ fuel assembly with eight grids (four mixing vane grids and four simple support grids). This research can provide best practice guidelines for the prediction of CHF in complex structures and the calculation results can provide a reference for the construction of large-scale CHF test benches and the location of the sensor installation.

2. Physical models

2.1. Governing equations

Compared with single-phase flow, mass, momentum, and energy transfer phenomena occur at the interface of two-phase flow. The control equations have been described in detail in our previous study [27]. In the Eulerian two-fluid model, each phase is treated as an interpenetrating continuum and the mass, momentum, and energy conservation equations of the liquid and vapor phases are solved.

2.2. Interfacial momentum transfer

The interaction force is usually divided into the drag force ($\vec{F}_D$) and the non-drag force. The non-drag force includes the lift force ($\vec{F}_L$), the turbulent dissipation force ($\vec{F}_{TD}$), the wall lubrication force ($\vec{F}_{WL}$), and the virtual mass force ($\vec{F}_{VM}$).

$$\vec{M}_k = \vec{F}_D + \vec{F}_{TD} + \vec{F}_L + \vec{F}_{WL} + \vec{F}_{VM}$$  \hspace{1cm} (1)

The common models of the various interphase forces are presented in Table 2. The lift force depends on the density of the continuous phase and the bubble diameter of the discrete phase [21]. The bubble diameter, however, is often smaller under higher pressure conditions; therefore, the lift force has little effect on the wall temperature distribution under high pressure conditions, which is ignored in this study. The application of lift and wall lubrication in adiabatic bubbly flow is relatively mature, but great uncertainties in the application of two-phase boiling flow [28].
virtual mass force needs to be considered when the discrete phase has an acceleration relative to the continuous phase, and it is often ignored in steady-state calculations. Because we conducted this study was conducted under high pressure conditions, we ignored the influences of the lift force, the wall lubrication force, and the virtual mass force in the calculations, and considered only the influence of the drag force and the turbulent dissipation force.

2.3. Wall boiling model

The wall boiling model for calculating the DNB phenomenon are based primarily on the wall boiling model proposed by Kurul and Podowski [34], which is also called the Rensselaer Polytechnic Institute (RPI) model. STAR-CCM+ extends the RPI model by considering the heat directly transferred from the heated wall to the vapor phase so that it can be used to calculate the CHF conditions. The total wall heat flux \( q_{\text{wall}} \) is divided into four components: the liquid phase convection heat flux \( q_l \), the evaporation heat flux \( q_e \), the quenching heat flux \( q_{\text{quench}} \), and the vapor phase convection heat flux \( q_g \), as follows:

\[
q_{\text{wall}} = \left( q_l + q_e + q_{\text{quench}} \right) \left( 1 - K_{\text{dry}} \right) + K_{\text{dry}} q_g
\]

where \( K_{\text{dry}} \) is the wall contact area fraction of the vapor. \( K_{\text{dry}} \) is estimated as follows:

\[
K_{\text{dry}} = \begin{cases} 
0, & \alpha_d < \alpha_{\text{dry}} \\
\beta^2 (3 - 2\beta)^2, & \alpha_d \geq \alpha_{\text{dry}} 
\end{cases}
\]

where

\[
\beta = \frac{\alpha_d - \alpha_{\text{dry}}}{\alpha_{\text{dry}}}
\]

where \( \alpha_d \) is the vapor volume fraction averaged over the bubbly layer thickness and \( \alpha_{\text{dry}} \) is the critical void fraction. According to a theoretical derivation, Weisman and Pei determined that the critical value of the void fraction was 0.82 [35]. The four components of the heat flux can be calculated using the following equations:

\[
q_e = \frac{\pi D_d^3}{6} \beta N_{\text{ws}} h_g
\]

and

\[
q_g = \frac{\beta C_{pg} U_{ch}}{g} (T_w - T_g)
\]

where \( U_{ch} \) and \( U_{*g} \) are the velocity of the liquid and vapor close to the heating wall, respectively; \( T_w \) and \( T_g \) are the dimensionless parameters of the liquid and vapor, respectively; \( C_{pg} \) and \( C_{pg} \) are the specific heats of the heat and vapor, respectively; \( h_g \) is the latent heat of vaporization; \( D_d \) and \( f \) are the bubble detachment diameter and the bubble detachment frequency, respectively; and \( N_{\text{ws}} \) is the nucleation site density; \( K_{\text{quench}} \) is the bubble influence wall area fraction, which is calculated using the following equation:

\[
K_{\text{quench}} = F_A \frac{\pi D_d^2}{4} N_{\text{ws}}
\]

where \( F_A \) is an area coefficient for scaling between the nucleation site area density and the wall area fraction of the bubble-induced quenching influences. The default value of \( F_A \) is 2.

2.4. Closure models

The closure models used in this study are presented in Table 3. In this study, we used the nucleation site density model developed by Li et al. and the model equation is as follows [15]:

\[
N_{\text{ws}} = N_0 (1 - \cos \theta) \exp (f(p)) \Delta T_{\text{sup}}^{-1/2}
\]

where:

\[
f(p) = 26.006 - 3.678 \exp (-2p) - 21.907 \exp \left( \frac{-p}{24.065} \right)
\]

and

\[
A = -0.0002p^2 + 0.0108p + 0.0119,
\]

\[
B = 0.122p + 1.988
\]

where \( p \) is the system pressure with the unit of MPa, and \( \Delta T_{\text{sup}} \) is
the wall superheat.

The Li model is similar to the Hibiki-Ishii nucleation site density model [36], which considers the influences of the pressure and the wall contact angle. The Li nucleation site density model also accounts for the influence of the wall superheat and is fitted based on a large number of experimental data. In addition, the Li model more easily obtains the obvious turning point in the boiling curve in the CHF calculations [15]. The Kocamustafaoğulları bubble departure diameter model based on the balance between gravity and surface tension is widely used to calculate flow boiling [37]. The frequency of bubble detachment is described by the Cole model [38].

2.5. Parameter sensitivity analysis

In the beginning of the simulation, this section makes a detailed sensitivity analysis of the grid size, the turbulence model and the interphase force model in the two-phase calculation. This study can provide a basis for the numerical simulation of subcooled flow boiling in fuel reactor assembly.

Bartolomei et al. carried out an experimental study on the subcooled flow boiling in a vertical tube [39]. To make a good comparison, experimental conditions under the pressure 4.5 MPa, heat flux 570 kW/m², inlet mass velocity 900 kg/(m²·s) and inlet subcooling temperature 60 K were selected as the inputs for this numerical study. In the experiment, the length of the heating section is 2 m and the diameter of the tube is 15.4 mm. The geometry of the numerical model is shown in Fig. 1.

2.5.1. Sensitivity analysis of grid and turbulence model

The height of the first layer of grid at the heating surface changes from 0.003 mm to 0.4 mm by setting different radial partition numbers. Under the current working conditions, the corresponding wall y⁺ range is 1–150. The grid parameters are set as shown in Table 4.

Five turbulence models including standard k-ε model, SST k-ω model, standard k-ε model, Realizable k-ε model and Realizable k-ε two-layer model were selected for sensitivity analysis. Different turbulence models need to cooperate with corresponding wall treatment methods. According to the different combinations of grid and turbulence model and matching with the corresponding wall treatment types, the table containing the computation examples is obtained as shown in Table 5.

Fig. 2 shows the change of void fraction as a function of thermodynamic quality under different grids solved by realizable k-ε model and SST k-ε model. The height of the first layer grid has a great influence on the calculation results. Mesh1 and Mesh2 have the largest number of grids, but a convergent solution is not obtained in the calculation using realizable k-ε model. It can be seen that for Euler two-fluid model, it is not the more grids divided in computing domain, the more accurate, the selected sized grids are suitable and exclude the dependence of the grid size.

Fig. 3 shows the void fraction and wall temperature distribution obtained by using different turbulence models under the Mesh5 grid size. It can be seen from the diagram that there is little difference between the calculated and experimental results of the wall temperature distribution of each turbulence model, and there is little difference between the calculated results of different turbulence models. The realizable k-ε model and the realizable k-ε two-layer model can well reflect the axial void fraction distribution.

2.5.2. Sensitivity analysis of boiling model

In order to analyze the sensitivity of the boiling model, the bubble nucleation site density model developed by Lemmert-Chawla (LC), Hibiki-Ishii (HI) and Li, the bubble detachment diameter model of Tolubinsky-Kostanchuk (TK) and Kocamustafaoğulları (KI) and the scaling coefficients of three bubble influence areas (1, 2 and 4) are combined to obtain the example table shown in Table 6.

As can be seen from Fig. 4, the scaling coefficient has little effect...
on the parameter distribution. Except for the combination of LC-KI models, the other five models can predict the wall temperature distribution better. Combined with the wall temperature and void fraction distribution curve, it can be concluded that the combination of HI-KI or Li-KI boiling model is recommended in the two-phase calculation.

2.5.3. Sensitivity analysis of interphase force model

In order to evaluate the influence of interphase forces on the calculation results, the example table in Table 7 is obtained by combining different non-drag models under the same drag model (Tomiyama). The turbulent dissipation force is Burns model, the wall lubrication force is Antal model, the virtual mass force is Auton model, and the lift model is Tomiyama model. Then, the effects of

Table 5

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Mesh1</th>
<th>Mesh2</th>
<th>Mesh3</th>
<th>Mesh4</th>
<th>Mesh5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard k-ω model</td>
<td>Low/Full</td>
<td>Low/Full</td>
<td>High/Full</td>
<td>High/Full</td>
<td>High/Full</td>
</tr>
<tr>
<td>SST k-ω model</td>
<td>Low/Full</td>
<td>Low/Full</td>
<td>High/Full</td>
<td>High/Full</td>
<td>High/Full</td>
</tr>
<tr>
<td>Realizable k-ε two-layer model</td>
<td>Full</td>
<td>Full</td>
<td>Full</td>
<td>Full</td>
<td>Full</td>
</tr>
<tr>
<td>Realizable k-ε model</td>
<td>High</td>
<td>High</td>
<td>High</td>
<td>High</td>
<td>High</td>
</tr>
<tr>
<td>Standard k-ε model</td>
<td>High</td>
<td>High</td>
<td>High</td>
<td>High</td>
<td>High</td>
</tr>
</tbody>
</table>

Table 6

<table>
<thead>
<tr>
<th>Example number</th>
<th>Nucleation site density model</th>
<th>Bubble detachment diameter model</th>
<th>Bubble influence area scaling factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td>HI</td>
<td>KI</td>
<td>1</td>
</tr>
<tr>
<td>C2</td>
<td>HI</td>
<td>KI</td>
<td>2</td>
</tr>
<tr>
<td>C3</td>
<td>HI</td>
<td>KI</td>
<td>4</td>
</tr>
<tr>
<td>C4</td>
<td>HI</td>
<td>TK</td>
<td>1</td>
</tr>
<tr>
<td>C5</td>
<td>HI</td>
<td>TK</td>
<td>2</td>
</tr>
<tr>
<td>C6</td>
<td>HI</td>
<td>TK</td>
<td>4</td>
</tr>
<tr>
<td>C7</td>
<td>LC</td>
<td>KI</td>
<td>2</td>
</tr>
<tr>
<td>C8</td>
<td>LC</td>
<td>TK</td>
<td>2</td>
</tr>
<tr>
<td>C9</td>
<td>Li</td>
<td>KI</td>
<td>2</td>
</tr>
<tr>
<td>C10</td>
<td>Li</td>
<td>TK</td>
<td>2</td>
</tr>
</tbody>
</table>

Fig. 2. Void fraction under each grids solved by different turbulence models.

Fig. 3. Calculation results of different turbulence models under Mesh5.
seven drag models on the calculation results are compared.

Fig. 5 shows the effects of different non-drag models on the radial void fraction and the axial wall temperature distribution. As can be seen from Fig. 5, when the non-drag force is not taken into account, the bubbles will concentrate near the heated surface, which will cause the wall temperature to rise. With the increase of turbulent dissipation force, the radial void fraction shows a flattening trend. The increase of lift will shift the peak position of the void fraction from the wall to the mainstream. However, the virtual mass force and wall lubrication force have little influence on the radial void fraction distribution. According to the above analysis, when using the Euler two-fluid model, the better results can be obtained by adding the influence of turbulent dissipative force and lift. However, due to the muddy effect of MVG in the fuel assembly, the influence of lift on the calculation results will be weakened. Therefore, only the turbulent dissipative force is taken into account in the later calculation.

Fig. 6 shows the calculation results using different drag models. For the current working conditions, each drag model can better reflect the distribution of axial void fraction, and the calculated results of Tomiyama drag model and symmetrical drag model are in good agreement with the experimental values.

3. CHF experimental data

We performed the CHF experiments in a 5 × 5 rod bundle under various test conditions at Nuclear Power Institute of China (NPIC). The 5 × 5 test bundle was equipped with eight spacer grids (four sets of mixing vane grids and four sets of simple support grids without mixing vanes) along the heated length. The layout of the grids and the rod configuration of the 5 × 5 test bundle in the experiment are shown in Fig. 7. The rod bundles simulated a Pressurized Water Reactors (PWR) fuel assembly with nine high-power (hot) rods at central locations and 16 low-power (cold) rods at peripheral locations inside a square frame of 66.1 mm in width. Each rod had an outer diameter of 9.5 mm. The spacing between rods and the frame was 3.1 mm (i.e., the rod pitch was 12.6 mm). The arrangement of rod radial peaking factors and the main dimensions of the 5 × 5 test bundle is illustrated in Fig. 8.

We selected 11 working conditions under different pressure, inlet subcooling, and mass flux ($P = 14.0–16.0$ MPa, $T_{sub} = 78–198$ K, $G = 2000–4000$ kg/(m²s)) from the 5 × 5 rod bundle CHF experiments. Details of the cases selected from the CHF experiments in this study are listed in Table 8. The uncertainty of the main parameters of the experiment is shown in Table 9.

4. Mesh and boundary conditions

We adopted the hybrid grid scheme to generate the mesh of the fuel bundle. The spacer regions in the 5 × 5 rod bundle were meshed using the polyhedral mesh, which is suitable for complicated structure, and the non-spacer regions were meshed using the swept mesh, which is suitable for a high-aspect-ratio geometry. We performed mesh refinement on small faces, such as the mixing vanes and strips. Five sets of meshes with different base size for the mixing vane grid (MVG) region (contains upstream and downstream areas) were generated for the mesh independence test. We finally selected the mesh base size corresponding to Mesh3 to generate the independent mesh in the 5 × 5 rod bundle after checking the pressure drop presented in Table 10. The final...
generated mesh contains 11.68 million cells and the typical mesh is shown in Fig. 9. The boundary conditions are listed in Table 11. We calculated the inlet velocity according to the ratio of mass flux to the density of the water. We set the exit of the fuel bundles as pressure outlet boundary. No-slip wall and adiabatic conditions were imposed on the square channel walls.

5. Results and discussion

5.1. Prediction method of CHF and its locations

Similar to the experimental process, we used the power step method to control the wall heat flux of the fuel rods in the numerical calculation. A steady simulation was carried out at the initial wall heat flux, and the changes of the wall superheat and the maximum void fraction of the vapor were monitored. Then the wall heat flux increased after the monitored parameters were stable and the calculation was convergent. We repeated the process to increase the wall heat flux until the CHF could be detected. Similar to the method used by Q. Li and Sablikova et al., the boiling curves was plotted to determine the occurrence of DNB [20,40]. Because of the effect of the latent heat of vaporization in the phase transition process, the superheat of the wall in the subcooled boiling state increased in a linear trend with the increase of the wall heat flux approximately. When the heat flux increased to a certain value, the superheat of the wall increased rapidly. We took Case 1 and Case 10 as examples to illustrate. As shown in Fig. 10, the heat flux corresponding to the deviation of the wall superheat from the linear increase was regarded as CHF.

Because the fuel rods were uniformly heated, the DNB would occur in the last MVG region (contains upstream and downstream
areas). We determined the specific location of DNB by identifying the area with the highest surface temperature and the highest void fraction. We considered two typical working conditions, Case 4 and Case 10, as examples to introduce the method to determine the location of DNB. The wall temperature and void fraction distribution in the last MVG region in Case 4 and Case 10 were extracted as shown in Fig. 11. We adjusted the display of the scalar field to a suitable range to obtain the position of the highest wall temperature and void fraction. The displayed rod bundle temperature was limited to 1–2 K below the maximum temperature. The void fraction on the surface of the rod bundle was larger than 0.9, which was the most likely location for the heat transfer deterioration.

The maximum temperature distribution of Case 4 was in two positions. One was located at the end of the heating section near the outlet of the fluid region, and the other was located in the MVG grid region. In contrast, the highest temperature of Case 10 was concentrated at the end of the heating section. The temperature of the 16 low-power rods at peripheral locations was lower than the nine high-power rods at the central locations. The region with the highest void fraction distribution was similar to that of the temperature distribution, but there were some differences. As shown in Fig. 11, the void fraction accumulated at the two side channels. The heat flux of the low-power rod, however, was lower than that of the central rod, so the temperature of the side channel was lower than that of the central channel. On the basis of the surface temperature of the rod bundle and the contours of void fraction distribution, it was evident that DNB was most likely to occur in the MVG or its downstream region under the Case 4 condition (corresponding axial position was 2.155–2.42 m). Under the Case 10 condition, the DNB first appeared at 2.28 m and its downstream region, namely, the region with the highest wall temperature and the largest void fraction.

To investigate the prediction accuracy, a ratio is defined as follows:

\[ R = \frac{CHF_{\text{calc}}}{CHF_{\text{exp}}} \]  

When \( R \) was equal to 1, the best prediction was obtained. We used \( R_V \) to denote the ratio of CHF values and \( R_L \) to represent the ratio of locations where CHF occurs. The calculated \( R_V \) and \( R_L \) under different conditions are shown in Fig. 12 and Fig. 13.

Fig. 12 shows the comparison of the experimental and predicted CHF values. As shown in Fig. 12, the deviation between the predicted value and the experimental value for all working conditions was within 25%. Moreover, the CHF value was underestimated in all numerical calculations, which could result in a conservative estimate when designing the maximum power of a fuel assembly. All working conditions were under high pressure (14.5–16.5 MPa), and the influence of pressure on the calculation was relatively small. To analyze the accuracy of the current numerical model in predicting the CHF value and the occurrence location of DNB under different boundary conditions, we divided all of the cases into two categories according to the different mass flux and inlet subcooling. The first group was the working condition with a low mass flux and high inlet subcooling, which included Case 3, Case 4, and Case 8 (G ≤ 2010 kg/m²s, \( T_{\text{sub}} ≥ 158K \)). The second group was the working condition with high mass flux and low inlet subcooling (G > 3000 kg/m²s, \( T_{\text{sub}} < 118K \)), which include the other eight cases. Fig. 12 shows that the accuracy of CHF prediction was affected significantly by inlet subcooling and mass flux. Under the condition of low mass flux and high inlet subcooling (i.e., the first group) the prediction of CHF value was in good agreement with the experimental value. On the contrary, under the condition of the second group, the prediction of the CHF value deviated significantly from the experimental value.

Fig. 13 shows the comparison of the experimental and predicted DNB locations. The deviation between the predicted value and

<table>
<thead>
<tr>
<th>Case No</th>
<th>P (MPa)</th>
<th>G (kg/(m²s))</th>
<th>T_{\text{sub}} (K)</th>
<th>CHF_{\text{exp}} (kW/m²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td>14.0</td>
<td>3000</td>
<td>90</td>
<td>2.56</td>
</tr>
<tr>
<td>C2</td>
<td>14.0</td>
<td>4000</td>
<td>79</td>
<td>2.89</td>
</tr>
<tr>
<td>C3</td>
<td>16.0</td>
<td>2000</td>
<td>198</td>
<td>2.89</td>
</tr>
<tr>
<td>C4</td>
<td>16.0</td>
<td>2000</td>
<td>159</td>
<td>2.55</td>
</tr>
<tr>
<td>C5</td>
<td>16.0</td>
<td>3000</td>
<td>118</td>
<td>2.88</td>
</tr>
<tr>
<td>C6</td>
<td>16.0</td>
<td>3000</td>
<td>84</td>
<td>2.42</td>
</tr>
<tr>
<td>C7</td>
<td>16.0</td>
<td>4000</td>
<td>78</td>
<td>2.86</td>
</tr>
<tr>
<td>C8</td>
<td>15.0</td>
<td>2000</td>
<td>180</td>
<td>2.74</td>
</tr>
<tr>
<td>C9</td>
<td>15.0</td>
<td>3000</td>
<td>116</td>
<td>2.88</td>
</tr>
<tr>
<td>C10</td>
<td>15.0</td>
<td>3500</td>
<td>95</td>
<td>2.87</td>
</tr>
<tr>
<td>C11</td>
<td>15.0</td>
<td>3500</td>
<td>78</td>
<td>2.63</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Pressure</th>
<th>Temperature</th>
<th>Mass flux</th>
<th>Critical heat flux</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unit</td>
<td>Pa</td>
<td>K</td>
<td>kg/(m²s)</td>
<td>W/m²</td>
</tr>
<tr>
<td>Uncertainty</td>
<td>0.87</td>
<td>0.79</td>
<td>1.30</td>
<td>2.67</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mesh No</th>
<th>Base Size (mm)</th>
<th>Mesh No</th>
<th>Pressure Drop (kPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh 1</td>
<td>1</td>
<td>3511160</td>
<td>19.096</td>
</tr>
<tr>
<td>Mesh 2</td>
<td>2</td>
<td>3218399</td>
<td>19.656</td>
</tr>
<tr>
<td>Mesh 3</td>
<td>3</td>
<td>3093688</td>
<td>19.692</td>
</tr>
<tr>
<td>Mesh 4</td>
<td>4</td>
<td>2186111</td>
<td>19.342</td>
</tr>
<tr>
<td>Mesh 5</td>
<td>5</td>
<td>1853400</td>
<td>19.105</td>
</tr>
</tbody>
</table>
the experimental value for most of the cases was within 10%. On the basis of the current numerical model, the predicted location of DNB was in good agreement with the experimental for the cases of high mass flux and low inlet subcooling (i.e., the second group), but there was a large deviation between the predicted location of DNB and the experimental value in the first group.

The prediction accuracy of the CFD method for the CHF value under high mass flux and low subcooling condition and DNB location under low mass flux and high subcooling condition still need further study (see Figs. 12 and 13).

5.2. Thermal-hydraulic characteristics in a 5 × 5 fuel assembly

We selected Case 4 and Case 10 as typical conditions to analyze the thermal-hydraulic characteristics, including temperature of liquid, pressure, and secondary flow distribution in the 5 × 5 fuel assembly when the DNB was detected. Then, based on Case 4, we introduced the flow field characteristics when DNB was detected.
Fig. 14 shows the cross-section average temperature variations along the flow direction under the two typical working conditions. The cross-section average temperatures increased gradually along the flow direction from the inlet, and the temperature at the outlet was still lower than the saturation temperature. The spacer grids had a certain influence on the coolant’s average temperature. The effect of MVG on coolant temperature was higher than that of the simple support grid (SSG). The mixing vanes on the MVG intensified the downstream flow field mixing and reduced the rate at which the coolant temperature rose.

Fig. 15 shows the cross-section average relative pressure (reference pressure relative to pressure outlet boundary) distribution along the flow direction under the two typical working conditions. The average relative pressure decreased along the flow direction. The total pressure drop of Case 10 in the condition of high mass flux was about 50 kPa, whereas the total pressure drop of Case 4 in the condition of low mass flux was about 16 kPa. The SSG and MVG grids reduced the flow area of the channel, causing significant pressure drops. The pressure drop of the MVG grids with a more complex structure was higher than that of the SSG grid. The pressure rose slightly after the coolant flowing out of the grids because of the increase in flow area of the channel.

One of the problems that fuel rod bundles face during operation is flow induced vibration. This flow-induced vibration is associated with the secondary flow. The secondary flow caused by the guiding role of the mixing vane leads to the considerably lateral velocity fluctuations of the coolant fluid [6]. In addition, secondary flow plays an important role in reducing the fuel rod surface temperature, reducing the average temperature of the rod bundle assembly, and enhancing the heat transfer efficiency. Fig. 16 shows the secondary flow profiles along the flow direction under the two typical working conditions. Both SSG and MVG grids significantly
increased the secondary flow. The mixing vanes in MVG grid played an important role in the downstream secondary flow distribution. After the coolant passed through the first SSG grid, the secondary flow decreased rapidly until it was negligible. After passing the second MVG grid, the flow field was strongly disturbed under the guidance of the mixing vanes, and this influence continued until the third grid (SSG). After the coolant passed through the third grid, the secondary flow was suppressed, which was significantly lower than that in the upstream of the grid. In addition, the secondary flow of high mass flux condition was higher, but the overall trend was consistent with that of the low mass flux condition.

Fig. 17 shows the void fraction distribution on the rod surface along the flow direction under Case 4. No obvious vapor was generated in the first half of the calculation domain. The void fraction increased gradually in the second half of the fluid domain. The reason for this gradual increase was that the coolant in the first half of the calculation domain was still in a state of high subcooling,
Fig. 17. Void fraction distribution on the rod surface.

Fig. 18. Contours of the different sections between -Dh and 20Dh in the last spacer grid.
and the heat emitted from the fuel rod was transferred by forced convection in the liquid. With the increase of the heat entering the coolant, the coolant temperature on the surface of the fuel rod in the second half of the calculation domain gradually increased to reach the saturation temperature, and then the subcooled boiling stage began. In addition, after the flow passed through the MVG grid, because of the mixing effect of the mixing vanes, the void fraction decreased rapidly and the distribution changed significantly, while the influence of the SSG grid on the flow field was relatively small and did not cause obvious changes.

For a thorough understanding of this process, we selected three characteristic sections of the last MVG grid area for qualitative analysis. Based on the root of MVG grid, we summarized the distributions of secondary flow, temperature, and void fraction at -Dh, 5 Dh, and 20 Dh (Fig. 18). Note that the secondary flow in the up-stream of the grid was relatively small, and the closer the nine high-power rods at central locations were, the higher the liquid temperature and void fraction were. After passing through the mixing vanes, the secondary flow increased rapidly and the mixing effect of the flow field was strengthened. The mixing of the vapor and the subcooled bulk flow intensified the condensation of the vapor, reduced the void fraction, and increased the temperature of the bulk region. When the flow was far away from the grid, the secondary flow decreased gradually while the void fraction increased again, and a large amount of vapor appeared on the surface of the 16 low-power rods at peripheral locations. DNB occurred at the location where the vapor accumulated and the surface temperature of the fuel rod was the highest.

6. Conclusion

We used the Eulerian two-fluid model combined with the extended wall boiling model to numerically simulate the 5 × 5 fuel rod bundle with spacer grids (four sets of mixing vane grids and four sets of simple support grids without mixing vanes). We calculated and analyzed 11 experimental conditions under different pressures, inlet temperatures, and mass flow rates. We confirmed the reliability of CFD analysis for the prediction of the CHF of the rod bundle channel and the boiling characteristics of the two-phase flow and analyzed the influence of the spacer grid and mixing vanes on the void fraction, temperature of the liquid, and the secondary flow distribution. The main conclusions of this study follow:

(1) The CFD method used in this study was shown to be in good agreement with the experimental results for predicting the CHF and DNB locations with the complex structure of the 5 × 5 rod bundle. The maximum deviation of CHF was less than 25%, and the maximum deviation of DNB location was less than 10%.

(2) Compared with the boundary pressure, inlet subcooling and mass flux had more significant effects on the accuracy of the CFD model. The prediction accuracy of the CFD method for CHF value under high mass flux and low subcooling condition and DNB location under low mass flux and high subcooling condition still needs further study.

(3) Both SSG and MVG grids can significantly increased the secondary flow. The mixing vanes in the MVG grid played an important role in the downstream secondary flow distribution. The flow field was strongly disturbed under the guidance of the mixing vanes, and the influence continued to exist until the next grid. The mixing of the vapor and the subcooled bulk flow intensified the condensation increased the vapor, reduced the void fraction and increased the temperature of the bulk region, which was beneficial for the enhancement of CHF in the rod channel.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References


