Numerical study on flow and heat transfer characteristics in the rod bundle channels under super critical pressure condition

Xingbo Yang, G.H. Su*, Wenxi Tian, Jiayun Wang, Suizheng Qiu

State Key Laboratory of Multiphase Flow in Power Engineering, Department of Nuclear Science and Technology, Xi’an Jiaotong University, Xi’an 710049, PR China

1. Introduction

The super critical water-cooled reactor (SCWR) is considered as one of the most promising Generation IV candidate reactors due to its simplified system, high thermal efficiency and sufficient operation experiences of the thermal power stations with super critical water-cooled cycle.

Several conceptual designs of SCWRs have been proposed: (a) super critical water-cooled thermal neutron reactor; (b) super critical water-cooled fast neutron reactor; (c) super critical water-cooled mixed neutron spectrum reactor; (d) super critical water-cooled pebble bed reactor; and (e) super critical heavy water-cooled reactor. Table 1 shows the detailed design parameters of present typical SCWRs around the world.

Although no boiling crisis occurs for the super critical coolant in the core, heat transfer deterioration occurring in the SCWR may lead to severe consequence. Therefore, the study of flow and heat transfer characteristics in the rod bundle channels in the super critical water-cooled reactor is of significant importance for its design and development.

The research on thermal hydraulic behavior of super critical fluids dates back to 1950s. Experimental and theoretical research on heat transfer under super critical pressure conditions was performed and was also conducted in the past decades (Jackson and Hall, 1979; Polyakov, 1991; Cheng and Schulenberg, 2001; Pioro and Duffey, 2005). In recent years, numerical simulation has been successfully adopted in the studies of heat transfer characteristics of the super critical water (Kim et al., 2004; Roelof and Komen, 2005; Oka et al., 2007; Cai et al., 2009). Compared with experimental investigation, numerical simulation costs much less and the results are also very instructive. Koshizuka et al. (1995) numerically analysed the deterioration in heat transfer at super critical water cooling in a vertical pipe. The results agreed with the experimental data of Yamagata et al. (1972). It was found that heat transfer deterioration is caused by two mechanisms depending on the mass flow rate. Yang et al. (2007) numerically investigated the heat transfer in upward flows of super critical water in circular tubes and in tight fuel rod bundles using the commercial CFD code. Some turbulence models were compared in the numerical simulations. The results were compared with experimental data and other heat transfer correlations in the super critical condition. It was found that there was a strong non-uniformity of the circumferential distribution of the cladding surface temperature in the square lattice bundle with a small pitch-to-diameter ratio (P/D), which did not occur in the triangular lattice bundle with a small P/D. This was caused by the large non-uniformity of the flow area in the cross section of sub-channels. Some improved suggestions were proposed to avoid the large circumferential temperature gradient at the cladding surface. Cheng et al. (2007) investigated the heat transfer of super critical water in various flow channels using the commercial CFD code software. Three different flow channels were selected and the impact of mesh structures, turbulence models, and flow channel configurations were analysed. The applicability of different turbulence models had been evaluated under super critical condition and a new definition for the onset of heat transfer deterioration was proposed. Shang et al. (2008) studied the system pressure effect on heat transfer of super critical water in
a horizontal round tube using CFD technique. It was found that when the buoyancy effect was negligible, the system pressure change had significant effects on the heat transfer. However, when the buoyancy effect was considerably strong, it had less effect due to the strong influences of the buoyancy force. Gu et al. (2008) numerically studied the thermal-hydraulic behavior of super critical water flows in two typical types of sub-channels in SCWR, i.e. square and triangular lattices. The results showed that the circumferential temperature distribution was non-uniform, especially when pitch-to-diameter ratio is small.

As described above, many numerical studies on the heat transfer characteristics of super critical fluids have been carried out. However, they were mostly limited to simple flow channel geometries. For the basic understanding of thermal-hydraulic behavior in SCWRs, in this study different turbulent models were evaluated, and the flow and heat transfer characteristics in different rod bundle channels were numerically investigated by using CFX codes.

2. Mathematical and physical models

The reactor considered in this study is one of the SCWRs proposed in China (Cheng, 2007) which is thermal and fast neutron mixed type. There are 100 fuel assemblies (FA) in the fast neutron region and 184 in the thermal neutron region.

Fig. 1 shows the detailed FA arrangement in the fast neutron region. The diameter of the fuel rod (D) is 8 mm. The gap between two rods (P) is 10.2 mm. Each fuel assembly consists of 289 fuel rods (17 × 17). The width of the FA is 177.2 mm. As shown in Fig. 1b, the red and blue regions are fuel and blanket layers, respectively. Each layer is 50 cm high and therefore the total height is 4.5 m. The fuel rod average linear power density is 16 kW/m.

Fig. 2 shows the cross section of the thermal neutron region. The diameter of the fuel rod (D) is 8 mm while the rod spacing (P) is 9.6 mm. The length of the rod bundle channel is 4 m, with 250 mm reflectors on both ends. Therefore its total length is 4.5 m. There are 180 fuel rods, including 24 burnable poison rods (yellow rods). The average wall heat flux is 600 kW/m².

The detailed parameters of the SCWR in China are given in Table 2.

2.1. Geometric structure and computational domain

Only a quarter of the fast and thermal FA is calculated taking symmetry into consideration. According to the geometric structures, the fast neutron channels are divided into three different kinds as shown in Fig. 3, i.e. the typical quadrilateral, the wall and the corner channels. The thermal neutron channels also include wall and corner channels but there are two different kinds of wall channels, as shown in Fig. 4. Only the shadow regions are calculated for symmetry. Furthermore, as the shadow regions are symmetric along the dashed lines, only half needs to be calculated.

2.2. Selection of turbulent models

The proper choice of the turbulent models is important for scientific calculation and can lead to satisfactory simulation results. Therefore, lots of research has been done on this field. Kim et al. (2004) applied more than 10 first order closure turbulence models as shown in Table 3 and found that the low-Reynolds number models were not able to estimate the wall temperature correctly. The RNG k – ԑ model with the enhanced near-wall treatment obtained the most satisfactory prediction. Other researchers (Cheng et al., 2007; Gu et al., 2007) investigated the heat transfer characteristics of SCWR in different flow channels and found that the second order turbulence models, i.e. the Reynolds stress model of Speziale (SSG) and the Reynolds stress model of Launder (LRR) give excellent prediction. Yang et al. (2007) found that the two-layer model is more accurate to predict the heat transfer at super critical pressure than other models using STAR-CD. However, due to the limitations of computational capabilities, they adopted standard high Reynolds number (Re) k – ԑ model with standard wall function to perform the CFD analyses. Gu et al. (2010) numerically investigated the heat transfer in sub-channels of a super fast reactor and found that the non-linear Speziale quadratic high Re k – ԑ model with two-layer near-wall treatment and the y+ value <1 give the acceptable results using STAR-CD.

Based on the review of the open literature on the turbulence models adopted under super critical pressure condition, different researchers have different conclusions. As there were no experimental data of bundle rod channels, most results were compared with the data of circular tubes and it’s hard to tell which model can simulate the turbulence flow best.

The RNG method has been employed in the turbulent flow field since 1970s. In late 1980s, Yakhok and Orazag (1986) theoretically deduced RNG k – ԑ turbulent model. They believed that turbulent flow in the inertia sub-region can be described by the N–S equations with random forces. Therefore, the N–S equations restricted by the boundary and initial conditions turn into the unrestricted Fourier transforming (Kandikar, 2002) so as to get rid of the high frequency and small size turbulent terms. Smith and Reynolds (1992) discovered the error in RNG k – ԑ turbulent model and deduced a modified one. Table 4 presents the comparison between the three models. Speziale et al. (1991) calculated the distribution of turbulent kinetic energy by using the original RNG k – ԑ model

### Table 1
Typical SCWRs around the world (Li and Wang, 2006).

<table>
<thead>
<tr>
<th>Name</th>
<th>Proposed by</th>
<th>Design concept</th>
<th>Moderate</th>
<th>Rated power (MW)</th>
<th>Outlet temperature (°C)</th>
<th>Pressure (MPa)</th>
<th>Net efficiency (%)</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>TWG1</td>
<td>Water TWG (Japan)</td>
<td>Fast neutron spectrum SCWR</td>
<td></td>
<td>D₂O</td>
<td>1728</td>
<td>Alternative</td>
<td>Alternative</td>
<td>38–45</td>
</tr>
<tr>
<td>W6–1</td>
<td>AECL (Canada)</td>
<td>CANDU-X-MARK1</td>
<td></td>
<td></td>
<td>910</td>
<td>430</td>
<td>25</td>
<td>44</td>
</tr>
<tr>
<td>W6–2</td>
<td>CANDU-XNC</td>
<td></td>
<td></td>
<td></td>
<td>370</td>
<td>400</td>
<td>25</td>
<td>41</td>
</tr>
<tr>
<td>W6–3</td>
<td>CANDU-ALX1</td>
<td></td>
<td></td>
<td></td>
<td>950</td>
<td>450</td>
<td>25</td>
<td>40.6</td>
</tr>
<tr>
<td>W6–4</td>
<td>CANDU-ALX2</td>
<td></td>
<td></td>
<td></td>
<td>1143</td>
<td>650</td>
<td>25</td>
<td>45</td>
</tr>
<tr>
<td>TWG2</td>
<td>– Europe</td>
<td>HFLWR</td>
<td></td>
<td>H₂O</td>
<td>1000</td>
<td>500</td>
<td>25</td>
<td>44</td>
</tr>
<tr>
<td>TWG3</td>
<td>– INEL (USA)</td>
<td>Thermal neutron spectrum SCWR</td>
<td></td>
<td>H₂O</td>
<td>1600</td>
<td>500</td>
<td>25</td>
<td>44</td>
</tr>
<tr>
<td>B5005KD1</td>
<td>Russia</td>
<td>Integrative SCWR</td>
<td></td>
<td>H₂O</td>
<td>515</td>
<td>381</td>
<td>23.6</td>
<td>38</td>
</tr>
</tbody>
</table>

1 For interpretation of color in Figs. 1–2 and 5–24, the reader is referred to the web version of this article.
and found the results were much larger than those by Large Eddy Simulation (LES), even worse than the typical $k - \varepsilon$ model. This is because the coefficient $C_{e1}$ (1.063) is less than that in the typical $k - \varepsilon$ model. In the uniform shear flow, the turbulent kinetic energy and its dissipation ratio grow exponentially as shown below:

$$k' \propto \exp(\lambda'), \varepsilon' \propto \exp(\lambda')$$

where,

$$\lambda = \frac{C_{pl}(C_{e2} - C_{e1})}{(C_{e1} - 1)(C_{e2} - 1)}^{0.5} \quad (2)$$

Therefore, when $C_{e1} = 1$, $\lambda$ becomes infinity. In the original RNG $k - \varepsilon$ model, $C_{e1} = 1.063$. This results in the very high eddy viscosity and turbulent kinetic growing ratio.

From Table 4 it can be seen that in the modified RNG $k - \varepsilon$ model, as $C_{e1}$ is 1.42, the overestimation of eddy viscosity is avoided. Therefore in this study the modified RNG $k - \varepsilon$ model is adopted to simulate the turbulent flow.

The governing equations (ANSYS CFX User Manual, 2006) are as follows:

Mass conservation equation:

$$\frac{\partial (\rho u_i)}{\partial x_i} = 0$$ \quad (3)

Momentum conservation equation:

$$\frac{\partial}{\partial x_j} \left( \rho u_i u_j \right) = \frac{\partial}{\partial x_j} \left[ \mu_{eff} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu_{eff} \frac{\partial u_i}{\partial x_k} \right] - \frac{\partial p}{\partial x_i} + \rho g_i$$ \quad (4)

Energy conservation equation:

$$\frac{\partial \left( \rho u_i c_p T \right)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \mu_{eff} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu_{eff} \frac{\partial u_i}{\partial x_k} \right]$$

$$+ \frac{\partial u_i}{\partial x_j} \left[ f \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{2}{3} \mu_{eff} \frac{\partial u_i}{\partial x_k} \right]$$ \quad (5)

RNG $k - \varepsilon$ model is adopted to close the turbulent kinetic energy and dissipation equations.

Turbulent kinetic energy equation:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_j} = \frac{\partial (\alpha_k \mu_{eff} \partial k / \partial x_j)}{\partial x_j} + C_k \alpha_k - \rho \varepsilon$$ \quad (6)
where $G_b$ is the effect of buoyancy on turbulent kinetic energy, and is defined as:

$$G_b = -g \frac{1}{\rho} \frac{\mu}{Pr} \left( \frac{\partial T}{\partial x} \right)_p$$  \hspace{1cm} (7)

where $G_k$ is turbulent kinetic energy produced by velocity gradient.

$$S = \sqrt{2S_p S_q} \frac{[\partial u_x/\partial x + \partial u_y/\partial y]}{2}$$  \hspace{1cm} (9)

Turbulent kinetic energy dissipation ratio:

$$\partial (\rho e)/\partial t + \partial (\rho u_i e)/\partial x_j = \partial (\rho \epsilon) / \partial x_j + C_{e1} \frac{e}{k} (G_k + C_{e2} G_b)$$

$$\quad + C_{e2} G_b - C_{e2} \rho \frac{e^2}{k} - R_e$$  \hspace{1cm} (10)

$$\mu_{mol} = \mu_{mol} \left[ 1 + \frac{C_{\mu}}{C_{\mu}} \left( \frac{k}{\rho \mu} \right)^{1/2} \right]$$  \hspace{1cm} (11)

where $\mu_{mol}$ is the molecule viscosity.

$$R_e = \frac{C_p \rho h}{1 + \eta / \eta_0} \frac{1}{k^2}$$  \hspace{1cm} (12)

where $\eta = Sk$, $\eta_0 = 4.38$, and $\beta = 0.012$.

The properties of the super critical water are calculated based on the IAPWS-IF97 correlations.

2.3. Meshing independence analysis

The grid number is very large because high density grid is necessary to accurately simulate the abrupt variation of the thermophysical properties in the super critical condition. In the super critical region, the dimensionless parameter $y^+$ should be smaller than 1 as too large $y^+$ can result in less accuracy (Koshizuka et al., 1995; Cheng et al., 2007).
Fig. 5 shows the detailed grid graph. It can be seen that the mesh is highly refined. The first node layer is only 0.002 mm from the wall, with the expansion coefficient ratio 1.2, and the maximum node gap is 0.1 mm. O-hexahedral mesh is adopted. The minimum and maximum grid qualities are 0.752 and 0.998, respectively, which indicates the present generated grid quality is quite good.

The boundary conditions are chosen based on the SCWR model proposed by Cheng (2007). The inlet Re (50,000–100,000) and temperature (300°C) are given. The fixed bilaterally heating wall heat flux is 600 kW/m². The outlet static pressure is 25 MPa. It is supposed that the proliferative layer in the fast neutron region is adiabatic. The gravity is considered and the flow directions of the fast and thermal neutron regions are upward and downward, respectively.

The meshing independence analysis is shown in Table 5. From Fig. 6 it can be seen that the difference between wall temperatures obtained by Mesh 2 and 3 is very small, i.e. <0.22%. The grid-independent solution is obtained. Therefore, Mesh 2 is the best and can be adopted in this simulation.

Fig. 7 shows the comparison between our calculation data and the experimental data (Yamagata et al., 1972). The heat transfer coefficient distribution in the super critical pressure region is calculated using both $k – \varepsilon$ and RNG $k – \varepsilon$ models. It can be seen the results agree well and the maximum relative error is <20%. The RNG model is better than the $k – \varepsilon$ model.

### 3. Results and discussion

Figs. 8 and 9 show the velocity and temperature distribution in the outlet section of the fast neutron region. It can be seen that the maximum velocity occurs in the typical quadrilateral sub-channels and the velocity difference between each typical quadrilateral sub-channel is small. The velocity near the wall is much smaller. As a result, the temperature near the wall is higher, and in the typical quadrilateral sub-channels is lower. The highest temperature occurs in the corner region.

Table 5

<table>
<thead>
<tr>
<th></th>
<th>Mesh 1</th>
<th>Mesh 2</th>
<th>Mesh 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node ($10^6$)</td>
<td>23</td>
<td>46</td>
<td>72</td>
</tr>
<tr>
<td>$y^+$</td>
<td>20</td>
<td>1</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Fig. 10 shows the lateral velocity distribution near the pseudocritical point. It can be seen that the lateral velocity near the wall is larger. Away from the wall, its value is smaller. The average velocity is about 8 m/s, which means the lateral oscillation is about 0.03%.

#### 3.1. Typical quadrilateral channels

Fig. 11 shows the lateral velocity distribution near the pseudocritical point under different $P/D$ values. It can be seen that the secondary flow direction is along the wall from $\theta = 0^\circ$ to $\theta = 45^\circ$, and the velocity increases first and then decreases. With the increase of Re, the lateral velocity decreases and the location of the maximum velocity moves. However, the trend of its moving direction is not very clear. When the Re and the gap between the wall and rod are fixed, with the increase of $P/D$, the lateral velocity also decreases and the location of the maximum velocity moves from the symmetric center to the wall.

Fig. 12 shows the wall temperature distribution for $\theta = 0^\circ$ and $\theta = 45^\circ$. The wall temperature increases with the main stream enthalpy and the increase is quite significant after the pseudo-critical point. When the main stream enthalpy reaches about 1500 kJ kg⁻¹, the wall temperature increases very slowly. For main stream en-
thalpy value of $>2200 \text{ kJ kg}^{-1}$, different trends appear depending on $P/D$ value. If $P/D$ is small (1.2, 1.3, and 1.4), the wall temperature still increases very slowly. But if $P/D$ is large (1.5 and 1.6), the wall temperature increases rapidly. If the main stream enthalpy and Re are fixed, with the increase of $P/D$, the wall temperature increases for $\theta = 45^\circ$. For $\theta = 0^\circ$, similar trend also exists except for the $P/D$ values of 1.3 and 1.2. This is because larger $P/D$ results in lower velocity, leading to the increase in local temperatures, sometimes even higher than the pseudo-critical point temperature. Therefore, the heat transfer deterioration (HTD) occurs and the wall temperature increases sharply.

Fig. 13 shows the heat transfer coefficient curves for different $P/D$ and main stream enthalpy when $\theta = 0^\circ$ and $\theta = 45^\circ$. It can be seen that the heat transfer coefficient increases with the main stream enthalpy, reaches the maximum value near the pseudo-critical point, and then decreases. With the increase of $P/D$, the heat transfer coefficient decreases and the location of the maximum heat transfer coefficient moves towards the lower enthalpy region.
The critical value for $P/D$ is 1.4. When $P/D$ is large (1.5 and 1.6), as the Reynolds Number is fixed, the flow velocity is relatively small. Before the main stream enthalpy reaches a high value, HTD already occurs. This phenomenon is also observed by other researchers.
(Koshizuka et al., 1995). When the velocity is small, the buoyancy force accelerates the flow velocity of the fluid in the near-wall region. This makes the flow velocity distribution to be flat and generation of turbulence energy is reduced. Thus the heat transfer is deteriorated. When \( P/D \) is smaller than 1.4, as the Reynolds Number is fixed, the flow velocity is relatively large. What’s more, the effect of \( P/D \) on the amplitude of turbulent mixing is relatively large (Gu et al., 2010). Therefore, the heat transfer condition is better and the main stream enthalpy can reach a high value. When \( P/D \) is 1.4, the two phenomena mentioned above influence the heat transfer characteristics at the same time. This causes the “flat region” at the maximum value.

Fig. 12 shows the wall temperature and heat transfer coefficient distribution in the same channel with different main stream enthalpy. It can be seen that the wall temperature for \( \theta = 0^\circ \) is higher than that of \( \theta = 45^\circ \). However, for the heat transfer coefficient the trend is opposite, except for a particular region. It can be seen that in the middle enthalpy region, when \( H_b \) is about 1500–1900 kJ kg\(^{-1}\), the HTC of \( \theta = 0^\circ \) is higher than that of \( 45^\circ \).

Fig. 13 shows the circumferential wall temperature and heat transfer coefficient distributions near the pseudo-critical point. When \( P/D \) is small (1.2 and 1.3), the heat transfer coefficient increases with \( \theta \). When \( P/D \) is large, with the increase of \( \theta \), the heat transfer coefficient increases first and then decreases. Heat transfer coefficient is observed to decrease with \( P/D \). As the \( P/D \) decreases, the fluid is heated by the wall and the viscosity increases locally in the near-wall region, which makes the viscous sub-layer thicker and the Pr number smaller. Both effects reduce the heat transfer. Therefore, the wall temperature shows the corresponding trends. It can be seen that with the \( P/D \) decrease, the circumferential non-uniformity increases.

3.2. Wall channels in fast neutron regions

Fig. 16 shows the velocity and temperature distributions near the pseudo-critical point in the wall channel 1 in the fast neutron region. It can be seen that the maximum velocity occurs in the middle of the rod bundle channel. The maximum temperature occurs in the region between the rod and wall, where the temperature gradient is very large.

Fig. 17 shows the inner wall temperature and heat transfer coefficient distribution. The wall temperature increases with the increase of the main stream enthalpy. The value first increases rapidly, then slowly and finally rapidly. If the rod diameter and the gap size between the rod and wall are fixed, with the increase of the inter-rod spacing, the temperature difference between the wall and main stream increases, and the heat transfer coefficient decreases. Furthermore, the locations of the maximum value of the heat transfer coefficient move towards the low enthalpy region with the \( P/D \) decreasing.
Fig. 18 represents the circumferential wall temperature and heat transfer coefficient distributions near the pseudo-critical point in wall channel 1. In the region near the wall, the smaller gap, larger viscosity resistance and lower velocity result in higher wall temperature. The velocity in the middle of the channel is larger and the wall temperature is lower. The circumferential temperature and heat transfer coefficient gradients on the rod surface are very large. With the increase of $P/D$ and the gap size between the rod and the wall fixed, the wall temperature increases significantly, where $h=0$. This is also consistent with the phenomenon mentioned above.

Fig. 19 represents the circumferential wall temperature distributions of the rod at different axial locations. It can be seen that near the pseudo-critical point, the circumferential temperature distribution is uniform, while it is non-uniform in the sub-critical and super critical regions. In the super critical region the non-uniformity is much more severe. The reason is intensive turbulence caused by abrupt thermophysical property changes near pseudo-critical point.

3.3. Wall channels in thermal neutron regions

Fig. 20 represents the circumferential velocity and temperature distributions near the pseudo-critical point in the wall channel 2 for the thermal neutron region. There are two velocity peaks in the channel. The velocity near the wall and HTC are smaller. Therefore the HTD occurs and wall temperature increases dramatically.
Fig. 21 shows the circumferential wall temperature distributions of the rod at different axial locations, which is similar to those in other channels. Near the pseudo-critical point, the circumferential temperature distribution is uniform, while in the sub-critical and super critical regions the distribution is non-uniform. There are two circumferential heat transfer coefficient peaks for $\theta = 45^\circ$ and $\theta = 135^\circ$.

3.4. Corner channels

Fig. 22 shows the temperature and velocity distributions near the pseudo-critical point in the corner channel. The velocity in the middle of the channel is larger and the heat transfer coefficient near the wall is lowest.

Fig. 23 represents the distributions of the inner wall temperature and heat transfer coefficient along the flow direction. In the sub-critical and super critical regions, the wall temperature increases fast. Near the pseudo-critical point this increase is much slower due to the higher heat transfer coefficient.

Fig. 24 shows the circumferential distributions of the wall temperature and heat transfer coefficient at different axial locations. It can be seen that near the pseudo-critical point, the circumferential temperature difference is small, while in the sub-critical and super critical regions it is large. The non-uniformity at the outlet is largest. Axially, in the middle of the channel, the fluid reaches the pseudo-critical point in the near-wall region, which causes the heat transfer coefficient to decrease and the wall temperature to increase rapidly.
4. Conclusions

In present study, numerical study was performed on the rod bundle channels based on one of the SCWRs proposed in China by using CFX codes. Different turbulent models were evaluated and the flow and heat transfer characteristics in some typical channels were investigated. The following conclusions have been obtained.

(1) In the typical quadrilateral channel, the effect of $P/D$ on HTC was studied. When $P/D < 1.4$, HTC increase with $P/D$ first and then decreases significantly. When $P/D = 1.4$, there is a “flat region” at the maximum value. If it is larger than 1.4, HTD occurs when the main stream enthalpy is still very small and more than one peak occurs.

(2) In the wall channel for the fast neutron region, the maximum velocity occurs in the middle of the rod bundle channel. The maximum temperature occurs between the rod and wall, where the temperature gradient is very large. With the increase of $P/D$, the temperature difference between the wall and main stream increases, and HTC decreases. Furthermore, the locations of the maximum value of the HTC move towards the low enthalpy region with the $P/D$ decreasing. In the wall channel 2 for the thermal neutron region near the pseudo-critical point, there are two circumferential HTC peaks, where $\theta = 45^\circ$ and $\theta = 135^\circ$. The maximum circumferential temperature occurs between the rod and wall.

(3) In the corner channel near the pseudo-critical point, the circumferential temperature difference is small, while in the sub-critical and super critical regions it is large. The non-uniformity at the outlet is largest. In the middle height of the channel, the fluid reaches the pseudo-critical point in the near-wall region, which causes the heat transfer coefficient to decrease and the wall temperature increases rapidly.

There are still a lot to do to improve the accuracy and reliability of CFD method and better models are needed to describe the flow and heat transfer characteristics for the fluid in super critical conditions in the future. Experimental data to verify the simulations is also necessary. Anyway, the study on SCWR is a hot issue nowadays and much more extensive research need to be done.

References


