Analysis of the flow distribution and mixing characteristics in the reactor pressure vessel

L.L. Tong a,⁎, L.Q. Hou b, X.W. Cao a

a School of Mechanical Engineering, Shanghai Jiao Tong University, Shanghai, 200240, PR China
b Nuclear Power Institute of China, Chengdu, 610041, PR China

A R T I C L E   I N F O

Article history:
Received 31 March 2020
Received in revised form 5 June 2020
Accepted 3 July 2020
Available online 21 August 2020

Keywords:
Flow distribution
Coolant mixing coefficient
Core inlet

A B S T R A C T

The analysis of the fluid flow characteristics in reactor pressure vessel is an important part of the hydraulic design of nuclear power plant, which is related to the structure design of reactor internals, the flow distribution at core inlet and the safety of nuclear power plant. The flow distribution and mixing characteristics in the pressurized reactor vessel for the 1000MWe advanced pressurized water reactor is analyzed by using Computational Fluid Dynamics (CFD) method in this study. The geometry model of the full-sized reactor vessel is built, which includes the cold and hot legs, downcomer, lower plenum, core, upper plenum, top plenum, and is verified with some parameters in DCD. Under normal condition, it is found that the flow skirt, core plate holes and outlet pipe cause pressure loss. The maximum and minimum flow coefficient is 1.028 and 0.961 respectively, and the standard deviation is 0.019. Compared with other reactor type, it shows relatively uniform of the flow distribution at the core inlet. The coolant mixing coefficient is investigated with adding additional variables, showing that mass transfer of coolant occurs near the interface. The coolant mainly distributes in the 90° area of the corresponding core inlet, and mixes at the interface with the coolant from the adjacent cold leg. 0.1% of corresponding coolant is still distributed at the inlet of the outer-ring components, indicating wide range of mixing coefficient distribution.

© 2020 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

The analysis of the fluid flow characteristics in the reactor pressure vessel is an important part of the hydraulic design of the nuclear power plant, which is related to the structure design of the reactor internals, the flow distribution at the core inlet and the safety of the nuclear power plant. Also, the better understanding on the coolant flow field in the reactor pressure vessel is the important foundation of establishing digital reactor system. The 1000MWe advanced pressurized water reactor contains lots of improvements, while also, whose hydraulic features are worth further investigating and validating.

Many researches focus on fluid flow and mixing characteristics in the lower plenum of reactor pressure vessel, including experiments and simulation analyses. For many years, experiments and numerical studies have been carried out on the reactor cooling system under normal operations or assumed accidents. In order to examine coolant mixing in the RPV of a German-type PWR, the ROCOM mixing experiments are performed, in which a 1:5 scaled Konvoi-type PWR test facility is built [1]. The obtained data could help clarify the mixing mechanism and provide data basis for the validation of CFD specifications [2,3]. The effect of density difference on the mixing and flow of water in the lower chamber of the pressurized water reactor (ROCOM) is studied by using CFX-5 and Trio U software [4]. The core inlet flow rates and exit pressure distributions of an APR+ (Advanced Power Reactor Plus) reactor are evaluated experimentally with the ACOP (APR + Core Flow & Pressure) test facility constructed with a linear reduced scale of 1/5 referring to the APR + reactor [5], in which 4-pump balanced flow conditions were conducted to examine the hydraulic characteristics of the reactor coolant flow.

CFD method can be used to simulate any part of fluid domain, including lower plenum and core, upper plenum interior, and upper plenum. Due to the difficulty to establish the complex geometry model in the core region, most scholars only study the local flow effect in the reactor vessel. The flow behavior in the downcomer and the lower plenum has a significant effect on the coolant distribution.
and mixing at core inlet [6,7], and the structure design of the fuel rod bundle and spacer grid determine the heat transfer capability in the core [8,9]. For AP1000 reactor, the Direct Vessel Injection (DVI) design will make it possible to rapidly cool the pressure vessel by injecting cooling water, but will cause a Pressurized Thermal Shock (PTS) phenomenon, threatening the integrity of the pressure vessel [10]. The decrease in the number of outlet pipes accelerates the PTS phenomenon, threatening the integrity of the pressure vessel injecting cooling water, but will cause a Pressurized Thermal Shock design will make it possible to rapidly cool the pressure vessel by locations, the study of local flow field cannot reflect the overall flow characteristics in the pressure vessel. It is necessary to embody the whole simplified pressure vessel fluid domain [13] to simulate the actual operation conditions. Using porous media instead of core region structure to simulate pressure drop and fluid velocity allows us to study the whole fluid domain [14].

From these studies, it can be inferred that with the normal operation of AP1000 in China, the analysis of flow distribution and mixing characteristics in AP1000 reactor vessel is very useful. It can provide detailed information or analysis method for the optimization or new design.

2. Computational model

2.1. Geometry model

The full-scaled reactor model consists two loops, which are composed of four cold legs (inlet pipes) and two hot legs (outlet pipes), as shown in Fig. 1. The mainstream fluid domains include cold and hot legs, downcomer, lower plenum with support structures, core region, upper plenum, and top plenum. Actual reactor structure is complicate, for reducing the unnecessary computation, the geometry model is simplified by follows. Under the condition of maintaining the mainstream, the core region is replaced by porous medium, and the top and bottom of the core are neglected. The bypass channels at the core barrel and hot leg are not considered in this operation condition, while four simplified cylindrical flow channels are established for simulating the effect of the bypass in the top plenum. Small size control rods in the upper plenum are numerous while have obvious block effect on coolant flow and control rods are replaced by the equivalent hexagonal prisms to reduce generating grids.

2.2. Mesh generation scheme

2.2.1. Mesh sensitivity analysis for local components

In order to obtain a tetrahedral grid with high quality, small quantity, and good calculation performance, which can reflect the geometric and fluid characteristics, a variety of dimensional functions are used for mesh generation to adapt to the internal components of various geometric shapes, mainly involving Curvature Size Function, Proximity Size Function, and Uniform Size Function in software Workbench Meshing.

Through the analysis of the local geometry, it is found that a small arc needs a small normal angle of mesh curvature, and when the arc is large, too many meshes will be generated. The hexagonal prism geometry is used to replace the cylinder control rod similar to the micro size circle to reduce unnecessary mesh generation. As shown in Fig. 2, a cylindrical rod with 18° curvature angle is needed to make the mesh completely close to the geometry boundary, which results in the aspect ratio of the mesh and a large number of meshes. However, this kind of grid is not conducive to capture the flow characteristics of small-scale flow field. The hexagonal rod only needs 36° or more curvature angle, which can make the mesh approximate the geometric characteristics under the condition of less mesh generation, indicating that the mesh generation using the equivalent geometric format can improve the adaptability of the mesh. The results of flow field calculation from the two grid schemes are in the acceptable range, shown in Fig. 2.

In order to achieve the computational independence of the mesh model for the important local components, the inlet and lower plenum, core region and upper subchannel are selected. Three order of magnitude meshes for each component are generated, as shown in Table 1. Fig. 3 shows the sampling lines of fluid domain. Considering the importance of flow field analysis, the velocity and pressure values of the main flow field under different grid density are selected for comparison, and the grid size sensitivity analysis is carried out under the same working condition. It can be seen that the number of medium grids meet the requirement of grid scheme independence, shown in Fig. 4.

2.2.2. Mesh generation of the full-scaled reactor model

The full-scale reactor model is composed with cold and hot legs, downcomer, lower plenum with support structures, core region, upper plenum, and top plenum. Tetrahedral unstructured grids are generated in the cold and hot legs, downcomer, the lower plenum, the upper plenum, and the top plenum fluid domain. The boundary layer is treated by the wall function method. Inflation method is used to generate the prism layer divided into three layers for the boundary layer of the downcomer, the lower plenum, the upper plenum, and the top plenum. The micro size structure such as control rod is simplified by the hexaprism control rod. Core structure is relatively simple and regular, and the hexahedral grid is adopted.

The full mesh model consist of 81.44 million tetrahedral cells and 1 million hexahedral cells, shown in Fig. 5. The quality of the generated grid model is evaluated using the method of grid comprehensive quality evaluation. The average mesh quality of tetrahedral element is above 0.71, the quality at the inlet and lower plenum is 0.79, the quality of hexahedral element is equal to 1.

![Fig. 1. Geometry model of the full-scaled reactor.](image-url)
2.3. Boundary conditions

Under normal operating conditions, the core coolant flow rate is relatively stable and the best estimated flow rate is adopted. The physical parameters of IAWPS database [15] are used, and the physical model is regarded as single-phase flow. At 280.4 °C, the

<table>
<thead>
<tr>
<th>Grid Size</th>
<th>Coarse (Million)</th>
<th>Medium (Million)</th>
<th>Fine (Million)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet and Lower Plenum</td>
<td>0.7</td>
<td>3.3</td>
<td>5.0</td>
</tr>
<tr>
<td>Core Region</td>
<td>0.08</td>
<td>0.25</td>
<td>0.50</td>
</tr>
<tr>
<td>Upper Subchannel</td>
<td>0.31</td>
<td>0.94</td>
<td>1.57</td>
</tr>
</tbody>
</table>

Fig. 2. Mesh generation of subchannel.

Fig. 3. Sampling lines of fluid domain.
loop flowrate at the boundary of the inlet pipe is 157500 m³/h, which is converted into a normal velocity of 20.26 m/s. The reference pressure is 15.499 MPa (2248 psi), and the relative pressure at the outlet pipe is set as 0 Pa. The standard $k$-$\varepsilon$ turbulence model and upwind discrete scheme are used to solve the flow distribution in the flow field. $Y$-plus value, which is very effective parameter for the standard $k$-$\varepsilon$ turbulence model, applies the range of $60 < y^+ < 300$. In theory, the first layer grid nodes are arranged in the logarithmic law layer, which can meet the accuracy requirements of the wall function. All walls are considered as smooth insulation wall without sliding. The convergence residual standard is set to $10^{-4}$. A 28-cores, 256G workstation is used for the calculation.

The source term of momentum loss of porous media is added to simulate the pressure drop of the core. Directional porous model is applied to the present numerical study, due to the main pressure drop along with $z$ direction (gravitational direction). The momentum source terms due to the pressure loss of porosity of the

![Inlet and Lower Plenum](image1)

![Core Region](image2)

![Upper Subchannel](image3)

**Fig. 4.** Results of mesh sensitivity analysis.
core can be described as equation (1), where $K_{\text{perm}}$ and $K_{\text{loss}}$ are the permeation rate and square loss coefficient of flow direction. The square resistance coefficient is set at 6700 kg m$^{-4}$, which is independent of the lateral flow behavior in the core and core porosity is 0.539.

$$S_{M,x} = -\frac{\mu}{K_{\text{perm}}}u_x - \frac{K_{\text{loss}}}{2} \frac{\rho}{H} |u_x|^2$$  

(1)

Table 2: Comparison of computational parameters.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Calculation</th>
<th>DCD [16]</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Core Pressure Drop (kPa)</td>
<td>274.7</td>
<td>275.1 ± 27.6</td>
<td>0.15%</td>
</tr>
<tr>
<td>Vessel Pressure Drop (kPa)</td>
<td>453.6</td>
<td>429.5 ± 42.7</td>
<td>5.6%</td>
</tr>
<tr>
<td>Head Cooling Flow (Percent of Total Flow)</td>
<td>1.95</td>
<td>2.0 ± 0.5</td>
<td>2.5%</td>
</tr>
<tr>
<td>Mass Flow Rate at Core (kg/s)</td>
<td>$1.46 \times 10^4$</td>
<td>$1.43 \times 10^4$</td>
<td>2.10%</td>
</tr>
</tbody>
</table>

Fig. 5. Mesh model of the full-scaled reactor.

Fig. 6. Flow distribution on the typical line in the RPV

a. Pressure and Velocity along the Centerline at $Z=6.2$ m in Downcomer

b. Pressure and Velocity along the Overall Streamline
3. Results and analysis

3.1. Verification of calculated results

The final CFD model of vessel is calculated and the results of hydraulic parameters are compared with the predicted values in DCD [16]. Shown from Table 2, the calculated core flow is slightly larger than the predicted value in DCD, and this may be due to the ideal assumption of normal flow at the inlet, but in fact, the coolant flow in the inlet pipe has a certain degree of turbulence. Consequently, the higher flow rate brings larger pressure loss of vessel, but all the flow parameters still meet the limitation of difference.

3.2. Flow field feature in reactor vessel

As shown in Fig. 6a, along the circumferential centerline at \( Z = 6.2 \text{ m} \) in the downcomer, the pressure and velocity values fluctuate periodically and are symmetrical at \( 0^\circ \) (positive direction of \( X \)) and \( 90^\circ, 180^\circ \) and \( 270^\circ \) positions, which is consistent with the characteristics of the vessel structure. Fig. 6b shows the variation of flow parameters along with the overall streamline, where there are two sharp fluctuations in pressure and velocity, one at the flow skirt and the other at the core plate holes. The pressure drop is mainly caused by the lower plenum structure before entering the core. As the coolant accelerates near the outlet pipe, the pressure decreases.

As shown in Fig. 7, the coolant flows through cold leg and the annular downcomer to the lower plenum, then through the low-speed area, into the through-hole of the lower core support plate and into the core. The flow in the core is relatively stable, while the flow near the outlet pipe is turbulent and high-speed.

The structure of the lower plenum can collect the coolant, restrain the eddy current, and distribute the flow at the inlet of the core, where the flow field has a significant influence on the flow behavior in the core. It can be seen from Fig. 8a, the mainstream flows into the core support plate holes from the downcomer, and the coolant at lower layer of the plenum forms a large eddy current due to the sheer of the fluid, and its flow velocity is slow down. It is obvious that most of the turbulence may be constrained by the flow skirt, which will also bring obvious pressure loss.

There are a large number of components in the upper plenum, which may severely disturb and hinder the coolant flow inside the guide tubes. The upper plenum is also an important fluid channel at the outlet of RPV, and its flow field is directly related to the stability of coolant flow at the outlet of the loop. Fig. 8b shows that the internal pressure energy of the coolant can be used to overcome the irrecoverable pressure loss, and most of the energy is converted into the kinetic energy of the coolant, so as to move to the outlet pipe, resulting in the local higher lateral velocity.

3.3. Flow distribution at core inlet

The results of the core flow distribution play an important role in the core power design. In order to provide good conditions for core heat transfer, results of core flow distribution should be uniform. Therefore, the monitoring surface which addresses the coolant normal flow rate is set in the through-hole of the lower support plate. In general, the normalized flow coefficient is defined as follows.

\[
\phi_i = \frac{q_i}{\bar{q}} \quad (2)
\]

Hereby, \( q_i \) is the flow rate of single core channel, \( \bar{q} \) is the average flow rate of the core. There are 4 variable-diameter flow-hole channels corresponding to each single core channel. The sum of the average flow rates of the flow-hole coolant is the flow rate of the single core channel.

The normalized flow coefficient distribution at the core inlet is obtained, as shown in Fig. 9. The results show that the distribution of the coolant flow is relatively uniform, and the normalized flow coefficient is large in the inner region and small in the outer region. The maximum value of 1.028 is in the 2nd layer, and the minimum value of 0.961 is in the 7th layer.

The parameters of normalized flow coefficient distribution are shown in Table 3, in which the standard deviation is less than 0.02, indicating that the flow is relatively uniform. Influenced by the high pressure level in the center of the lower plenum, the average value of the \( 3 \times 3 \) region in the center is larger than the average value of the external region, which indicates that the coolant flow transfer occurs in the process of entering the lower core plate.

The normalized flow coefficient distribution at core inlet for CNP1000 is calculated [17], in which, the maximum value is 1.078,
and the minimum value is 0.909. The maximum and average difference between adjacent channels are much lower for AP1000, which indicates that the flow distribution in the core inlet of AP1000 is more uniform.

In order to further study the non-uniformity of the flow distribution, the non-uniform coefficient is defined as the value of flow deviation of the component sub-channel from the average value 1, as shown in Fig. 10. The coefficient value is about 4%, located in the 6th layer, and the maximum positive coolant offset is about 2.6%, located in the 1st layer. With the increase of the number of radial component layers, the distribution of non-uniformity coefficient increase gradually until the 7th layer. The uniformity of flow gradient between adjacent layers can be expressed by the extension of the reference line close to the uniform gradient. The more the data concentrates near the line, the more uniform the flow gradient distribution is. The coolant flow in the central region is evenly distributed with high flow rate, while the coolant flow in the external region fluctuates greatly with low flow rate.

3.4. Coolant mixing coefficient

Due to the convection interaction of the fluid, the coolant has certain mixing characteristics during the flow process, and the mass transfer between the coolants from the cold legs mainly occurs near
the interface. The additional variable is an inactive scalar component transmitted by the fluid. It is added in the cold pipe, and it diffuses into the core along with the coolant flow. Moreover, the monitoring surface is still set in the through-hole of the lower support plate. When the coolant from each cold leg is distributed in the lower core support plate, labels can be recognized.

The numerical calculation of the distribution of the additional variable is one of the effective methods to solve the fluid distribution. The general form of the additional variable transfer equation can be described as Equation (3) [15].

\[ \frac{\partial (\rho \phi)}{\partial t} + \nabla \cdot (\rho \mu \phi) = \nabla \cdot (\rho D_{\phi} \nabla \phi) + S_{\phi} \]  

(3)

Where, \( \rho \) is the density of the fluid; \( \phi \) is conservation quantity per unit volume of additional variables, namely concentration; \( \phi \) is the Conserved quantities of additional variables per unit mass of fluid, \( \phi = \phi / \rho; \) \( \mu \) is the fluid velocity vector; \( D_{\phi} \) is the diffusion coefficient of the additional variable, which is the function of the fluid velocity and viscosity coefficient; \( S_{\phi} \) is the source term for the additional variable.

For the additional variable in turbulence, the solution is the transport equation of the Reynolds-Average model as Equation (4).

\[ \frac{\partial (\rho \phi)}{\partial t} + \nabla \cdot (\rho \mu \phi) = \nabla \cdot \left( \left( \rho D_{\phi} + \frac{\mu}{S_{\text{St}}} \right) \nabla \phi \right) + S_{\phi} \]  

(4)

Where, \( \mu_t \) is the turbulent viscosity, which is equal to the product of turbulent velocity and turbulent length scale in the two-equation mode; \( S_{\text{St}} \) is turbulent Schmidt number, which is equal to the ratio of turbulent viscosity and turbulent diffusion coefficient.

By adding additional variables at the cold pipe, the coolant from the four inlets is labeled and tracked for analysis. Figs. 11 and 12 gives the coolant mixing coefficient at the lower core support plate from the two cold legs. It is found that mass transfer of coolant occurs near the interface. The coolant mainly distributes in the 90° area of the corresponding core inlet, and mixes at the interface with the coolant from the adjacent cold leg. 0.1% of corresponding coolant is still distributed at the inlet of the outer-ring components, indicating wide range of mixing coefficient distribution.

### 4. Conclusions

A full-scale reactor model is built to study flow distribution at core inlet and coolant mixing coefficient for AP1000. It is found that the flow skirt, core plate holes and outlet pipe will bring pressure loss in the process of coolant flow. According the analysis results of normalized flow coefficient, it is found that the normalized flow coefficient is large in the inner region and small in the outer region, and the maximum value of 1.028 is in the 2nd layer, and the minimum value of 0.961 is in the 7th layer. Compared with that of CNP1000, the results show that the distribution of the coolant flow is relatively uniform, which can give some design modification of
internal structure. The coolant mixing coefficient investigated by adding additional variables has a wide range, and the mass transfer occurs near the interface mainly. The overall method can give some interesting reference for improved design.

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.

Appendix A. Supplementary data

Supplementary data to this article can be found online at https://doi.org/10.1016/j.net.2020.07.002.

References


[15] ANSYS CFX Help, Computational Fluid Dynamics, Release 18.2, ANSYS Inc. Copyright © 2011 Elsevier BV All rights reserved.
