The study of a reactor cooling pump under two-phase flow

P Wang, S Q Yuan, X L Wang and F Zhang
National research center of pumps, Jiangsu University, Zhenjiang, 212013, China

E-mail address: wangpeng2017@yahoo.com

Abstract. In this paper, the steady pressure field has been investigated numerically by computational fluid dynamics (CFD) in a nuclear reactor cooling pump. As a multiphase approach the Eulerian-Eulerian two fluid model has been applied to calculated five computational models with different kinds of blades. The analysis of inner flow field of the five model pumps shows that the pressure in the impeller increases with the increase of the gas contents and the pressure distributions are irregular at the inlet of different blades when the gas contents less than 20%. With the increase of the number of blades, the vortexes at the outlet of impeller decrease whereas the vortexes in the deep of the volute markedly increases and high velocity of the fluid huddle is generated gradually at the outlet pipes. Under the action of centrifugal force and Coriolis force, gas phase mainly concentrated at the lower velocity and lower pressure area. The radial force on the impeller gradually increases with the increase of the gas contents.

1. Introduction
Nuclear safety has been studied as one of the major issues since the inception of the nuclear industry and power generation. Establishing and maintaining core cooling to ensuring containment integrity are two main goals for nuclear safety [1-2]. As the heart of nuclear power plant, it is important for the performance of reactor cooling pump when under the two-phase flow. A.Poullikkas [3] has used the high-speed video(HSV) to observe the bubbles motion in the impeller passages and pointed out that the bubbles tend to concentrate at the blade suction side, the “high” gas contents(around 9 percent by volume) would form the passage blockage with a resulting break in the pump flow. Yuan [4] has investigated the collector discharge pressure fluctuation characteristics and the related factors in a nuclear coolant pump and pointed out the fluctuation amplitude is increased gradually from the top side wall of casing to the bottom, and the shaft rotating frequency is the major factor influencing pressure fluctuation at ever monitor when the pump is working at the design specifications.

Although a great deal of work related to traditional PWRs(Pressurized Water Reactor) has been done in the past few years[5-9], the mechanism of small break LOCA (Loss of Coolant Accident) in AP1000 has not been clearly understood, especially, the performance of coolant pump under two-phase flow(gas-liquid). To guarantee the safety operation of nuclear reactor, it is essential to study the inner gas-liquid flow in the impeller of cooling pump. In this paper, five different impellers are selected as the research subjects to investigate the performance of cooling pump under two-phase flow. The cooling pump performances and internal flow patterns are predicted by the commercial CFX 14.5. The main purpose of this paper is to detail description of the two phase flow in nuclear reactor cooling pump and discuss the relation between the gas contents and the force which load at the impeller and inner velocity flow patterns is also presented in this paper.
2. Numerical simulation methods

2.1. Model parameter and mesh generation

The design parameters are shown in Table 1.

| Parameter | Value  |
|-----------|--------|
| $Q$ (m$^3$/h) | 17886 |
| $H$ (m) | 111.3 |
| $n$ (r/min) | 1450 |
| Design pressure (MPa) | 17.1 |
| Specific Speed ($n_s$) | 344 |

According to the design parameters, the computational domains are created based on the pump geometries that include 5 following fluid domains: inlet section, impeller, guide vane, volute and outlet pipes. The inlet section and outlet are extended 5x their respective diameters so that the flow can be fully developed at the two sections. After modeling in Pro/E, the 3D geometry computational domains are shown as Figure 1. The structured grids for the computational domains are generated using the grid generation tool ANSYS ICEM CFD 14.5, the impeller without shroud are partially shown in Figure 2 and the computational domains are input to the flow solver, CFX 14.5, for simulation.

2.2. Basic assumptions

In this paper, the Eulerian-Eulerian two-phase fluid model is selected to emphasize the two aspects of the dynamic interactions between gas and liquid phases. Gas-liquid two phase flow is affected by many factors, the situation is complex. In order to simplify this research in this paper, a simplified physical model should be used. The following basic assumptions are put forward:

1) The flow pattern is regarded as steady flow, and the liquid phase is an incompressible fluid.
2) The structure of bubbles cannot be changed when they move into the flow passages and their diameter is far less than the diameter of the flow passages.
3) The gravity of gas-liquid two phase flow is ignored.

2.3. Governing equations

For incompressible bubbly flows without mass transfer, the instantaneous mass balance equations in the liquid and in the gas are respectively:

\[
\frac{\partial (1 - X_G) \rho_G}{\partial t} + \frac{\partial (1 - X_G) \rho_G u_{Gj}}{\partial x_j} = 0, \quad \frac{\partial X_G \rho_G}{\partial t} + \frac{\partial X_G \rho_G u_{Gj}}{\partial x_j} = 0
\]

where $u_{Gj}$, $\rho_G$, and $X_G$ are respectively the velocity, the density and the characteristic function of the
gas phase \((X_G = 1\) in the gas and 0 in the liquid) . To simplify, the subscript G is used for the gas phase and the subscript L is omitted for the liquid phase throughout the paper. Thus \(u, \rho\) and \((1 - X_G)\) are respectively the velocity, the density and the characteristic function of the liquid phase.

At each point where the liquid phase is present, the instantaneous momentum balance is:

\[
(1 - X_G) \left( \frac{D}{Dt} u_i = \frac{\partial}{\partial x_i} \sigma_{ij} + \rho g_i \right)
\]  

where \(\frac{D}{Dt} = \frac{\partial}{\partial t} + u_j \frac{\partial}{\partial x_j}\), \(\sigma_{ij}\) is the stress tensor and \(g_i\) is the acceleration of gravity.

For the specific case of bubbly flows, we neglect the acceleration and the weight of the gas as compared to the force exerted by the liquid on the bubbles because of the contrast of the densities \(\rho_G \ll \rho\); so, in each point where the gas phase is present, the instantaneous momentum equation in the gas indicates that the volume density of the total force \(f_{pi}\) exerted by the liquid on the bubbles is zero:

\[
X_G f_{pi} = 0
\]

The instantaneous momentum balance (3) represents an Eulerian description of the movement of the gas. It expresses the interfacial transfer as a local density of the force exerted by the liquid on the bubbles. In this model, the Reynolds stress tensor of the continuous phase is split into two parts, a turbulent dissipative part produced by the gradient of mean velocity and by the wakes of the bubbles and a pseudo-turbulent non-dissipative part induced by the displacements of the bubbles: each part is predetermined by a transport equation [10].

2.4. Numerical method and boundary conditions

The hydraulic passages of the modeled pump are taken in consideration as part of the computational domain. The pitch change for the interface between the rotor and stator is set as fixed rotor. The rotating speed is set as 1450rpm.

The normal velocity components are given at the inlet, and the outlet boundary is assumed to be pressure generated by the pump of the fluid. Standard wall function is adopted to predict the turbulent flow near-wall, and at the fluid physical surfaces of the pump were set to be no-slip wall, while the air physical surfaces were set to be free-slip wall. The turbulence is set to an initial intensity of 5\% at the inlets. Besides, the turbulence model can predict the development of bubbly flows well. The convergence precision is set as \(10^{-4}\).

3. Results and discussion

3.1. Internal pressure field analysis

Due to space limitations, we only present 4 kinds of different blades schemes to analysis, which are 5, 6, 8 and 9. The pressure field distributions for 4 different blades under the calculated conditions are shown in figure 3.

From the Figure.3a) we know that the lower pressure area mainly appeared at the inlet of the impeller and also a small amount of it can be seen at the outlet edge of impeller. With the increasing of the gas contents, the lower pressure increased from the pressure side to the suction side that is the gas mainly concentrated on the suction side of the impeller. From the Figure.3b) to Figure.3d), with the increasing of the blades, the lower pressure area spread from the pressure face to the suction face. When are 9 blades, as seen at Figure.3d), appeared a large amount of high pressure area at all blades near the inlet and outlet only Figure.3a) with 5 blades have lower and more even pressure area distribution. On the other hand, the pressure in the impeller increased with the increasing of the gas contents and the pressure distributions are irregularly at the inlet of different blades when the gas contents fewer than 20\%.
Figure 3. The static pressure distribution of different blades under different gas contents: a) 5 blades; b) 6 blades; c) 8 blades; d) 9 blades.

The distributions for gas contents of 25% are shown in Figure 3, here the high pressure area entirely fill the sides of the blades, and inevitably lower the pump performance. The main reason is probably because the larger the gas contents are, the lower inertia force of fluid has, can't keep the blades work better that poor the performance. Besides, with the increasing of blades, appeared a large amount of high pressure area that the losses within the impeller increases and the pump performance
Figure 4. Streamline distribution under cross section with different gas contents: a) 5 blades; b) 7 blades; c) 9 blades.
3.2. Cross section streamlines distribution analysis
Considering space limitations, the fluid streamline distribution of cross section for the pump with 5, 7 and 9 blades are shown in figure 4, it can be seen that the distribution of streamlines within the whole section are uneven. The high velocity can be seen in the impeller; however, large amount of vortexes appeared in depths of volute and accumulated a small amount of gas wake at the end of the diffuser.

With the increase of the blades number, the vortexes of the outlet of impeller decreased, whereas the vortexes in the deep of the volute markedly increased. At the same time, the high velocity of the fluid huddle at the outlet pipes gradually. With the increase of gas contents, the vortexes in the deep of volute are direct proportional to the gas contents gradually, that is the velocity fluctuation change in the direction of large scale and irregular. The main reason is probably because the higher the gas contents, the ability of blades to do work to the fluid is decrease, gas would change the flow trend near the outlet. Besides, due to the density of the gas phase is far less than the density of the liquid phase, under the action of centrifugal force and Coriolis force gets gas phase mainly concentrated at the lower velocity and lower pressure area.

3.3 The comparative analysis of the radial force of the impeller
The pulsation curves for the radial force of the impeller as the function of the rotation angle of the impeller in one cycle under the working conditions of five kinds of gas contents from 5% to 25% are shown in figure 5. In the figure, the rotation angle increasing from the zero to 360 degree and direction of the radial force, and the projection on one cycle of the impeller each point respectively represents the axial radial force of the impeller at the moment.

![Figure 5. The curves for different blades of the impeller radial force with the impeller rotation angle at different gas contents in one cycle.](image)

It can be seen that the radial force gradually increases with the increase of the gas contents, when the gas were 5%, the impeller with 6 and 7 blades bear the minimum force at all rotation. The force of 5, 8 and 9 blades, especially the 8 blades have the biggest force compared to the other schemes. When
compared with the list five schemes, it can be seen that the 5 blades with the minimum force at the suction side, whereas the other schemes have the biggest force. Local have obvious phenomenon of circle round on the blades, however, no matter at which side of the blades, the force increasing with the gas contents. Interesting, the force on the suction side with 9 blades is minimum when compared with all schemes.

4. Conclusions
Using Euler- Euler two-phase flow model, the reactor cooling pump under of different gas contents is simulated and the analysis results show that:

1) The pressure in the impeller increased with the increasing of the gas contents and the pressure distributions are unregularly at the inlet of different blades when the gas contents fewer than 20%, with the increasing of the blades, the lower pressure area spread from the pressure face to the suction face that the losses within the impeller increases and the pump performance drop.

2) The high velocity of the fluid huddle at the outlet pipes gradually, with the increase of the blades number, the vortexes of the outlet of impeller decreased, whereas the vortexes in the deep of the volute markedly increased, with the increase of gas contents, the velocity fluctuation change in the direction of large scale and irregular.

3) Compared with the list five schemes, it can be seen that the 5 blades with the minimum force at the suction side, whereas the other schemes have the biggest force and the radial force gradually increases with the increase of the gas contents.

Acknowledgments
This research was supported by the State Key Program of National Natural Science Foundation of China (Grant No. 51239005), the National Science & Technology Pillar Program of China (Grant No. 2011BAF14B04), the National Natural Science Foundation of China (Grant No. 51379091) and Technology Support Program, Priority Academic Program Development of Jiangsu Higher Education Institutions (PAPD)

References
[1] Furukawa A 1991 Z&E JSME Fall annual meeting B 165-7
[2] Yoshida Y 1991 JSME Fall annual meeting B 168-70
[3] Poullikkas A 2000 Progress in Nuclear Energy 36 123-30
[4] Yuan S Q, Zhu R S and Zheng B Y 2012 Atomic Energy Sci. and Techno. 46(10) 1202-6
[5] Su X S 2000 Foreign Nuclear Power 2000(6) 20-
[6] Poullikkas A 2000 Progress in Nuclear Energy 36 123-30
[7] Wang W W, Su G H, Qiu S Z and Tian W X 2011 Progress Nuclear Energy 53(4) 407-19
[8] Long Y, Zhu R S, Q Fu and Yuan S Q 2014 J. Drainage and Irrigation Machinery Eng. 32(4) 290-5
[9] Yang J, Wang W W, Qiu S Z, Tian W X and Su G H 2012 Annals of Nuclear Energy 46(4) 81-9
[10] Chahed J, Roig A and Masbernat L 2003 Int. J. Multiphase Flow 29(1) 23-49