Study on Flow Interference of Multi-group Parallel Jets between Heat Transfer Tubes

Weirang Wu*, Yuchun Tao, Tong Yu, Xunzong Yang and Chun Ye
Suzhou Nuclear Power Research Institute Co., Ltd. Jiangsu Suzhou, 215004, China

*Corresponding author email: wuweirang@cgnpc.com.cn

Abstract. In order to find an approach to optimize the hydraulic cleaning equipment of steam generator secondary side of CPR1000 reactor, the Computational Fluid Dynamics (CFD) method was used to simulate the flow of multiple groups of parallel jets. In this simulation, the RNG k-ε turbulence model and the gas-liquid two-phase flow model is employed to distribute the jet flow between heat transfer tubes. The results show that with the increase of the jet angle, the pressure aggregation on the tube sheet decreases obviously, which is caused by the deflection force due to the pressure difference between the two sides of the jet. With the jets distance increasing, the interaction between jets decreases obviously. When the distance is up to more than nine times of tube spacing, the force that causes deflection of the jet will almost disappear. With this distance, the jets can maintain a good linearity and obtain a satisfied cleaning effect on the heat transfer tubes.

1. Introduction
Steam generators with vertical U-shape tubes are commonly used in PWR nuclear power plants, to transfer the heat energy from the primary loop side in the nuclear reactor to the secondary loop, and produce water steam in the secondary loop side[1]. In the secondary loop, the surface of the tube sheet is usually covered by deposit of non-volatile impurities and corrosion products. The deposit will form a corrosive environment at the surface of heat transfer tube and the tube sheet, which will cause the corrosion of the heat transfer tube. In serious cases, it will cause radioactive leakage and thermal power loss. Therefore, during each shutdown and overhaul period, sludge lancing and video inspection on the secondary side of steam generator are pretty necessary[2].

The main cleaning methods of steam generator in nuclear power plant are mechanical cleaning and chemical cleaning. At present, mechanical hydraulic cleaning is mainly used to remove the deposited sludge on tube sheet with high pressure jet and circulating water loop system[3,4]. The spray gun cleans the surface of tube sheet hydraulically through handhole or eyehole. During the handhole cleaning, as shown in Figure 1(a), the handhole spray gun is introduced into the steam generator from the handhole at 0 or 180 degree position, and the actuator drives the spray gun to move step-by-step along the central passageway between the tubes, then rotates along its own axis to clean the outside surface of the heat transfer tube and the surface of the tube plate. At the same time, the circulating spray gun is loaded at the 0 or 180 degree position, and the cleaning water is sprayed along the loop passageway to drive the sludge which is expelled from the clearance between tubes. The sludge converged at the suction orifice at 90 degree and 270 degree position, which is sucked out from the suction pipeline to the filtering system to realize the water loop circulation. During eyehole cleaning, the high pressure spray gun and the circulating spray gun changes over, and the similar processes is performed, as shown in Figure 1(b).
There are 4474 heat transfer tubes in the steam generator, which are arranged in a square shape with only 8.38 mm visible clearance. The incident points of the spray gun installed at the handhole or the eyehole are all on the surface 350mm above of the tube sheet. The structure of the spray gun is shown in Figure 2. The nozzle of the handhole spray gun has an internal diameter of 1.7mm and a supply pressure of about 14.4Mpa, and the nozzle of the eyehole spray gun has an internal diameter of 1.4 mm and a supply pressure of 18.8Mpa.

When the spray gun is used to clean the tube sheet of the steam generator, multiple groups of parallel jets need to pass through the dense heat transfer tubes. Due to the different swing angles of the spray gun, the distance that jet needs to pass through between the heat transfer tubes is about 0.35m~1.6m. In order to ensure the cleaning effect of tube sheet, it is necessary to keep the linearity of the jet with high-pressure in the process of cleaning, to avoid the situation that most of the jet acts on the heat transfer tubes rather than reaching the surface of the tube sheet. Therefore, in the above range, whether the jet can maintain a satisfied linearity to reach the surface of the tube sheet is the key to achieve the desired cleaning effect of the mechanical hydraulic cleaning equipment.

Due to the characteristics of dense heat transfer tubes, it is difficult to achieve real-time video observation in the cleaning process, and it is impossible to grasp the parameters such as the shape and pressure of water jet. The simulation of jet flow field by CFD method can accurately and concretely obtain the jet shape and other key parameters[5-6]. Han[7] simulated two parallel water jets with a spacing of 17.8 mm and a width of 5.8 mm using Realizable k-ε and SST k-ω turbulence models. The adsorption phenomena and velocity distribution of the two jets were studied. Mukul[8] analyzed jet interference in water by LES large eddy simulation. Saya[9] experimentally studied the morphology of two parallel water jets with a distance of 12 mm and a diameter of 5.8 mm in the water using PIV, and also observed the obvious phenomenon of mutual adsorption. Yiming[10] carried out an experimental study on the jet morphology of 45 equal-distance parallel jets. It was found that the two jets adsorbed each other, and the jets near the edge adsorbed more easily by the jets in the central region.

In this paper, the CFD numerical simulation of the jet flow field in the process of hydraulic cleaning is carried out by using the method of gas-liquid two-phase flow, to analyze the shape of jet and the flow.
characteristics between heat transfer tubes. Because the process of handhole cleaning is similar to that of eyehole cleaning, this paper chooses the process of handhole cleaning as the object of simulation analysis.

2. Numerical Method
The computational region of handhole cleaning jet flow is shown in Figure 3. In order to set the rotation angle of the spray gun conveniently, the calculation area is divided into two parts: the gun region and jet region. The whole gun region is placed in the circular hole of the jet region. Fully structured grids are used in the gun region, and the quality of the grids is over 0.65. Because of playing a key role in the flow pattern of the subsequent high-speed jet, the grids of the nozzle part is necessary to be refined [11]. The number of grids in the gun region is determined to be 1.7 million, as shown in Figure 3. The grid was encrypted appropriately in order to capture the jet flow better. Local refinement was carried out at the interface of the two regions, and the number of grids in this area was determined to be 4.120 million.

![Figure 3. Computing domain grid structure](image)

To simulate the flow inside the two regions, the turbulence model RNG k-ε is utilized. This model is derived based on a rigorous statistical technique. RNG k-ε model introduces an analytically-derived differential formula for effective viscosity that accounts for low-Reynolds-number effects. The mathematical closure of RNG k-ε model is listed as follows:

\[
\frac{\partial \rho k}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \alpha_k \mu_{eff} \right) \frac{\partial k}{\partial x_j} \right] + G_b + G_b - \rho \varepsilon - Y_M
\]  

where \( k \) is the turbulent kinetic energy, \( \varepsilon \) is the turbulent dissipation rate. The effective viscosity and turbulent viscosity read as

\[
\mu_{eff} = \mu + \mu_t
\]

\[
\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}
\]

For the calculation of governing equations, SIMPLEC scheme is used. High Resolution Scheme is chosen as the discrete scheme of convection term. As a high order scheme, it can reduce the numerical diffusion and provide higher accuracy [12].

The working medium in the jet flow field is air and water at 25°C, and the reference pressure is a standard atmospheric pressure. The VOF (Volume of Fluid) model is used to separate gas phase and liquid phase, based on their volume fraction of total fluid. The velocity of inlet water from the four nozzles is set as 138 m/s according to the actual operation condition. The outlet is set as an open boundary with zero pressure gradient, which has good stability and convergence without setting the flow direction on the boundary [13].
3. Deflection of the Jet Flow

In high pressure cleaning, the surface cleaning effect is mainly affected by the water jet reaching the tube sheet. Therefore, it is necessary to perform a study of the forces on the tube sheet and the development of the jet flow during the rotation of the spray gun. Angles of 30, 36, 42, 48, 54 and 60 degrees are selected to describe the different jet flow character when the spray gun is rotating as shown in Figure 4, and the pressure distribution on the surface of the tube sheet at the above angles is shown in Figure 5. Combining with the two figures, it can be seen that when the jet angle is 30 degrees, the interaction between the four jets is weak, and the jets can all reach the tube sheet. The direction of jets at both sides deflect slightly which leading to a deviation of high pressure point on the tube sheet. When the jet angle is 36 degrees, the jets at both sides catch the heat transfer tubes due to the interaction between jets, which makes the pressure points on the tube sheet disperse. Only the two jets in the middle can still reach the tube sheet, and the high pressure points on the tube sheet are convergent. When the jet angle is 42 degrees, the two jets in the middle also catch the heat transfer tubes slightly, and then changes their direction and reach the bottom of the heat transfer tube. The high pressure point locates at the area between the two jets. When the jet angle is 48 degrees, the two jets at both sides gradually deflect to the space below the two jets in the middle. When the jet angle is 54 and 60 degrees, there is no obvious convergent pressure point on the tube sheet due to the four jets all crashes the tubes one after another.

![Figure 4. Streamline at different jet angles](image-url)
Figure 5. Pressure distribution on tube sheet at different jet angles

In order to reveal the reason of the deflection of the above-mentioned jets, two cross-sections as shown in Figure 6(a) are selected in the jet area when the jet angle is 30 degree, and the pressure contour of the selected two sections are extracted, as shown in Figure 6(b) and (c). It can be seen that the low pressure present in the area between each of the jets, which directly results in the pressure difference of the two jets on the periphery which leads to a force perpendicular to the direction of velocity on the water column. However each of the internal jets has a basically symmetrical pressure distribution on its two sides, and the force leading to the deflection of water column is not significant.

4. Effect of Distance

To find a way to improve the linearity of the jet, two jets through the tubes with 60 degree jet angle was simulated at several distances, as shown in Figure 7. When the jets are at a distance of 2 and 3 rows of tubes, the linearity of the jet does not change fundamentally. When the jet is at a distance of 6
rows of tubes, the jets can reach the tube sheet without touching the tubes. When the jet is at a distance of 9 rows of tubes, interaction between the two jets is almost eliminated.

(a) At the distance of 2 rows of tubes                         (b) At the distance of 3 rows of tubes
(c) At the distance of 6 rows of tubes                            (d) At the distance of 2 rows of tubes

Figure 7. Jet flow at different distance

Based on the above analysis, the distance between nozzles can be optimized to meet the requirements of distance to ensure the linearity. The spacing between nozzles should be as large as possible, and then improve the focus of pressure distribution on the tube plate.

5. Conclusion

In this paper, flow characters of parallel jets passing through heat transfer tubes are described by performing a CFD simulation. The mutual attraction between the jets is caused by the pressure difference on both sides of the jet, which causes the jet suffering a force perpendicular to its velocity direction and resulting in a jet deflection. The deflection of jet is not conducive to the linearity of jet and obstructs the jet reaching the tube sheet without touching tubes. Increasing the distance between the jets can effectively weaken the attraction effect between the jets. When the distance between the jets is large enough, the force causing deflection of the jets is eliminated to maintain the jet linearity.

References

[1] Ding X S and Li S J 1994 Mechanical cleaning of steam generator in PWR nuclear power plant Nuclear Power Engineering vol 15 pp 323-327
[2] Wu W R and Li J et al. 2015 Optimum design of recovery and filtration system for hydraulic cleaning equipment of steam generator Hydraulic and pneumatic vol 7 pp 72-73
[3] Ding X S 1999 Mechanical cleaning and chemical cleaning of steam generator in nuclear power plant Chemical cleaning vol 15 pp 20-26
[4] Tao Y C and Wu W R et al. 2016 Steam generator hard mud location distribution and flexible cleaning method Cleaning world vol 32 pp 36-39
[5] Xu R and Du C L et al. 2011 Jet simulation of nozzles with different contraction angles Mining machinery vol 39 pp 36-38
[6] Zhang Z and Zhang Q F 2013 Jet simulation of high pressure nozzles Mechanical and electrical engineering vol 30 pp 185-187
[7] Li H and Anand N et al. 2018 Computational study of turbulent flow interaction between twin rectangular jets International Journal of Heat & Mass Transfer vol 119 pp 752-767.
[8] Bisoi M and Das M K et al. 2017 Turbulent statistics in flow field due to interaction of two plane parallel jets Physics of Fluids vol 29 pp 125.
[9] Lee S and Hassan Y A 2017 Experimental study of flow structures near the merging point of two parallel plane jets using PIV&POD International Journal of Heat & Mass Transfer vol 16 pp 871-888
[10] Li Y and Li B et al. 2018 Flow and heat transfer of parallel multiple jets obliquely impinging on a flat surface Applied Thermal Engineering vol 133 pp 588-603
[11] Han L F and Guo J B et al. 2014 Hydraulic simulation and experimental study of centrifugal compressors Fluid machinery vol 42 pp 5-7
[12] Niu P and Sun Q G et al. 2013 Oil-gas two-phase jet simulation analysis based on FLUNT Information technology vol 41 pp 91-93
[13] Liu L F and Huang F et al. 2010 Flow field simulation of non-submerged pure water jet nozzle based on FLUENT Manufacturing technology and machine tool vol 10 pp 56-60