Numerical study on flow around vane in a channel using immersed boundary method

J F Huang1,3, S Yang1, J Yao2 and L X Zhang2
1College of Hydraulic Engineering, Yunnan Agricultural University, Kunming, Yunnan 650201, China
2Department of Engineering Mechanics, Kunming University of Science and Technology, Kunming, Yunnan 650051, China

E-mail: hjf30@126.com

Abstract. The immersed boundary method is applied to simulate the flow around a guide vane of hydro-turbine. The complex dynamic boundary effect caused by the moving guide vanes is modelled as the forcing term in the Navier-Stokes equation. Numerical simulation of three-dimensional transient turbulence is conducted to flow around one moving guide vane of a benchmark hydro-turbine unit in a channel. The dynamical Smagorinsky-Lilly model of large eddy simulation is also used. The evolution process of the dynamic vortex structure is obtained by the interaction between the guide vane and the flow around vane to reveal the physical mechanism of wake oscillation flow around vane. This method serves as a reference for the fine simulation and study of the flow around the multi-stage cascade flow in hydro-turbine. The research on the transient effects of the coupling upstream pipeline system and hydro-turbine has theoretical guiding significance.

1. Introduction
The fluid-structure interaction problem in hydro-turbine has been under the attention of engineering and academia in recent 20 years. With the characteristics of simple and fast power regulation, the hydro-turbine has a very important role in the regulation of the power system of the hydropower station. The guide vane of hydro-turbine is controlled by the wicket gates to regulate the flow rate of hydro-turbine, so as to change the power output of hydro-turbine. In the transition process of the hydropower station, the regulating movement of the moving guide vane of hydro-turbine will cause the pressure oscillation on the upstream pressure piping system, and the transient characteristics of the downstream hydro-turbine will be the main reason for the vibration and even stop of the machine. The regulation motion of the moving guide vane of hydro-turbine is a complex transient dynamic boundary fluid dynamics problem. Its accurate simulation is restricted by the computational fluid-structure coupling dynamic method and the computer hardware condition.

Influenced by free flow, the guide vane trailing edge will be dragged out of the wake when rotating the guide vane. The wake is merged into intense spiral Karman vortex, whose azimuth is changed instantaneously. The wake further changed the guide vane hydrodynamic and vibration response, thus produced important influence on performance and vibration amplitude of hydro-turbine. Therefore, the development of appropriate and accurate calculation method about the wake and its distribution is the foundation of the fluid-structure coupling dynamics. It is an important and difficult part of the hydro-turbine technology field. Numerical simulation of flow around airfoil with moving boundary is one of
the hot issues in recent years, because of its wide range of flow background and complex flow characteristics. The different numerical and experimental methods were used to study the flow characteristics of various airfoils by many scholars [1-3]. This paper attempts to develop a way to consider the interaction between the fluid and the guide vanes, and the interaction between the stay vanes and the guide vanes. However, because the fluid motion in the rotating machinery is quite complex, the simplified moving guide vane is classified as a guide vane in a channel. Firstly the flow around a guide vane in a channel is simulated, and then the dynamic flow around the double linear cascade in the channel is simulated. These works are the basis for the simulation of the whole hydro-turbine double cascade in the future.

The traditional grid generation technology can be divided into structured grid and unstructured grid. The characteristics of structured grid is directly in the near the object surface to generate orthogonal the body fitted grid (Body-fitted grid), then on the grid nodes solving control equations through coordinate transformation. The structure grid has the advantages of standard structured grid, and easy programming, which provides high precision simulation in solving simple regions. The drawback is that it is difficult for complex geometry to generate the body fitted grid. With a different structured grid, unstructured grid node with the adjacent grid has the same unit, so the finite control nodes of the body (finite control volume) can be used in arbitrary shape. Unstructured grid can deal with the complex geometry boundary problems. Due to the poor nodes correlation, the computation is complex, and its efficiency is low. In fact, only the 2D unstructured grid generation technology is relatively mature, but the 3D unstructured grid still faces many problems.

Regarding the moving boundary problems, the use of adaptive technology in structured and unstructured grid is a way to solve this kind of problem, but the grid needs to constantly generate new grid and refine the grid programming complexity, and thus the workload is very large and the loss of accuracy too [4]. Because each time-step needs recalculating by grid generation especially in the fluid-structure interaction problem, and the variables of the time layer is calculated by the interpolation of the flow field value at last time step. The force of the fluid and the solid is inconsistent with the real time layer, which results in a deviation from the actual physical. In recent years, a series of fixed Descartes Cartesian grid for solving complex boundary technology obtained the full attention and rapid development, especially in the so-called “immersed boundary method (IBM)”. The immersed boundary method first was proposed by professor C. S. Peskin of New York University mathematics in 1977 [5], and used to simulate the coupling mechanism between the blood flow and heart muscle. The biggest characteristic of this method is that the calculation is performed in a fixed orthogonal Descartes Cartesian grid, and Peskin presented a new method to deal with the influence on flow field of immersed boundary. Until recently, the immersed boundary method has developed many different modifications and improvements [6-19], and now the range of applications from the initial biomechanics extends to almost all areas of computational fluid mechanics [20-22]. In addition, the immersed boundary method is not only used for flow field calculation of fluid-solid boundary, also for flow of fluid-fluid and fluid-gas boundary calculation [23,24].

The immersed boundary method is the most critical of how the forcing term added to the governing equations. It is a sign of the distinction between different types of immersed boundary methods. In the governing equations, the forcing term can be applied in two ways: one is adding forcing term in the momentum equations, and another is adding forcing term in the continuity equation. According to the different processing methods of the forcing term, the immersed boundary method can be divided into two categories [25]: “continuous forcing approach” and “discrete forcing approach”. The forcing term for continuous forcing approach is incorporated into the continuous equations before discretization. An attractive feature of the continuous forcing approach is that it is formulated being independent of the underlying spatial discretization. The discretization of forcing source generally has analytical expression, which meets the requirements of some specific mechanical relationship type (such as Hooke’s law). It is mainly used to deal with elastic boundary problems. This approach has reliable physical foundation, and its implementation is very simple, which has been successfully applied to biological flow and multiphase flow problems. The forcing term for discrete forcing approach is
introduced after the equations are discretized. The discrete forcing approach depends on the discretization method. It generally cannot obtain the analytical expressions. It is mainly used for the treatment of solid interface problems (such as flow around a rigid body). Although it is not as good as continuous forcing approach, but it can be very good to describe the immersed boundary with high Reynolds number flows. This allows direct control over the numerical accuracy, stability, and discrete conservation properties of the solver.

In this paper, the immersed boundary method (IBM) is applied to simulate flow around vane, and cascade in hydro-turbine, combined with dynamic Smagorinsky-Lilly model of large eddy simulation (LES), numerical simulation of flow around one moving vane passage of the wicket gate in a benchmark hydro-turbine used in our laboratory at Kunming University of Science and Technology. The numerical results show that the propagation characteristics of blade airfoil wake, analysis of the unsteady flow around a vane, vortex structure development process and interaction.

2. The combined LES-IBM approach

Using the Gauss filter function, the original N-S equation through the filter processing to achieve the separation of large and small scale structures, and the introduction of the force source of the solid structure immersed boundary motion effect, flow control equations are:

\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} \left( \bar{u}_i \bar{u}_j \right) = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} + f_i, \tag{1}
\]

\[
\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{2}
\]

In the above equations, \(\bar{u}_i\) and \(\bar{u}_j\) is component of velocity. \(t\) is time. \(\rho\) is fluid density. \(\bar{p}\) is the pressure. \(\nu\) is the kinematic viscosity coefficient of fluid. \(f_i\) is the force source of the characterization of immersed boundary effect. The horizontal bar on the variable head indicates the variable after filtering. \(\tau_{ij}\) considering the effect of small-scale subgrid stress can be expressed as

\[
\tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \tag{3}
\]

\[- \left( \tau_{ij} - \frac{\delta_{ij}}{3} \tau_{kk} \right) = 2\nu_{sgs} \bar{S}_{ij} \tag{4}
\]

\[
\bar{S}_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \tag{5}
\]

\(\delta_{ij}\) is Kronecker constant. \(\tau_{kk}\) is viscous stress. \(\nu_{sgs}\) is the subgrid eddy viscosity coefficient. \(\bar{S}_{ij}\) is fluid strain rate tensor. In order to reflect the differences of flow characteristics in the guide vane closing movement process at different times, and conveniently combined with the existing solver, this paper uses dynamic subgrid stress on Smagorinsky-Lilly model. This model overcomes the shortcomings of the conventional Smagorinsky model. The subgrid scale eddy viscosity coefficient is

\[
\nu_{sgs} = C_d \Delta^3 \left| \bar{S}_{ij} \right|, \quad \text{and} \quad C_d \text{ is the coefficients of dynamic Smagorinsky-Lilly model with temporal and spatial variations. It is no longer a constant but with changes in flow coefficient model.} \ \Delta \text{ is the filtering characteristics of grid scale.}
\]

Discrete force method by handling regulating vane motion has been used to stimulate the effect of
immersed boundary. By adding a force body similar momentum source through the interface program, the flow field and the IBM problem of moving guide vane combined with the solver of ANSYS-CFX reached a complete solution. The solver will be updated automatically each time step the immersed boundary grid points from the beginning of the position, and then establish a series of internal nodes in the interior of immersed solid. In order to make the fluid velocity trend immersed boundary velocity solve will use force source to deal with fluid node in the interior of immersed solid.

Specifically, because there is no coupling relation between the flow field and the surface of the immersed boundary nodes, the information transfer between them requires an interpolation process. Peskin first proposed to use the Dirac function to construct the interpolation scheme, but proved its computational efficiency is low in the three-dimensional case. We use bilinear interpolation format to replace the Dirac function.

In the current study, the fluid velocities \( \mathbf{u}(X,t) \) are interpolated to a immersed boundary point, \( X = (X,Y) \) from the four surrounding grid points denoted by the indices \( (i, j) \), \( (i+1, j) \), \( (i, j+1) \) and \( (i+1, j+1) \), using bilinear interpolation,

\[
\mathbf{u}(X,t) = \sum_{i,j} D_{i,j}(\mathbf{X}) \mathbf{u}_{i,j}
\]

\[D_{i,j}(\mathbf{X}) = d(X-x_i)d(Y-y_j)\]

\[d(X-x_i) = \begin{cases} 
(X-x_{i+1})/(x_i-x_{i+1}) & \text{if } x_i < X \\
1 & \text{if } x_i = X \\
(X-x_{i-1})/(x_i-x_{i-1}) & \text{if } x_i > X
\end{cases}
\]

For the 3D case, eight nodes on the grid flow field surrounded by \( X \) interpolation, using the same format.

Obviously, this interpolation method is more simple and accurate than the Dirac function method. At the same time, bilinear interpolation is only needed to calculate the adjacent grid points without the need for full or partial numerical integration. With the increase of the dimension, the amount of computation is not increased significantly. Therefore, its computational approach is more efficient than that of the Dirac function method.

3. The IBM-LES simulation of dynamical flow around a moving guide vane in a channel

3.1. The establishment of computational model

For the calculation of one guide vane of the wicket gate in a benchmark hydro-turbine in our laboratory at Kunming University of Science and Technology and its closing movement of the representative, the type of hydro-turbine rated head 10 m, rated flow flux of 0.7 m\(^3\)/s, the rated speed of 600 r/min. The hydro-turbine and guide vane are shown in figures 1 and 2. In order to reveal formation and evolution mechanism of wake because of guide vane regulation, vane-channel model corresponding to one guide vane passage is built. One passage is expanded into a channel, and formed flow around one moving vane in a channel for guide vane regulation. The vane-channel model was shown in figure 3. The Reynolds number of calculation condition corresponding to one guide vane based on chord is 120421. The computational domain is \( 12L \times 4L \times 1.1L \), where \( L \) is chord length of one guide vane. In order to fully capture wake, the trailing edge of vane at streamwise direction is 8L.
After grid independence verification, the total grids numbers are 1,498,572. The vane region is divided by a hybrid grid, and the grid size is 3 mm. The nodes numbers are 10,870, and the grids numbers are 491,113. The fluid region is divided by hexahedron grid with eight nodes, and the grid size is 4 mm. The nodes numbers are 150,9872, and the grids numbers are 1,449,459. The computational grid is verified to be grid independently. For the interpolation of IBM, non-uniform mesh interpolation technique is used at the interface of fluid-solid. The dynamic Smagorinsky-Lilly model of large eddy simulation is used for the subgrid stress. The immersed boundary method is applied to deal with the moving boundary of vane. The fully implicit multi-grid coupled solution technology has achieved in accelerating the convergence.

![Figure 1. The hydro generator unit.](image1)

![Figure 2. Hydrofoil of the wicket gate.](image2)

![Figure 3. Vane-channel model used in computation.](image3)

![Figure 4. Curve of vane opening via flow rate.](image4)

3.2. **Boundary conditions**

Inlet: the actual unit vane opening corresponding to the change flow rate (show in figure 4), converted into one vane corresponding to the velocity, imposed on inlet as a known velocity boundary. Outlet:
the pressure outlet boundary condition. Assuming normalwise periodic boundary conditions and spanwise rigid non-slip wall, no-slip conditions are applied to the surface of vane.

The closing law of vane in hydro-turbine is divided into two straight line. The first of time is 5s, the second time is 10s, and total is 15s. Calculation time step is 0.001s. Using the computer as the 8 core CPU, and 24G memory workstation, one calculation takes about 360 hours.

3.3. Analysis of computational results

3.3.1. The changes of pressure in the process of moving vane. In the guide vane two segment closing process, the moving vane in a channel was simulated and recorded to the spanwise section (z=0.5 L) and normalwise section (y=2 L) on the change of pressure with time. Figure 5 shows the pressure distribution for several typical times in the spanwise section (z=0.5 L) and normalwise section (y=2 L). As can be seen in close action guide vane rotation process, the whole flow field pressure distribution becomes non-uniform and asymmetric, the upstream region variable for positive pressure area and downstream region variable for negative pressure zone, pressure distribution in a channel with variation of time showed a complex unsteady evolution process. At t=1s (vane rotates 9°), the small range high pressure region is found in leading edge of vane on spanwise section due to flow impact, and the negative pressure center appears at the bottom of vane. The negative pressure zone emerges near the upper and lower surface of the vane trailing edge. There are two negative pressure zones around the negative force surface of the vane on normalwise section. At t=3s (vane rotates 27°), the negative pressure center of vane is merged into a circular area, and a small range of negative pressure center surrounding the zones. The flow separation points near the negative force surface of vane formation gradually developed into vortex. With the development of the flow, the vortices begin to move away from the negative force surface of the guide vane, which influence the flow field in the downstream zones. At the t=5s (vane rotates 45°), a long narrow negative pressure zone is formed behind the vane on spanwise section. The distance between a strong negative pressure center and the vane trailing edge is very close, a strong negative pressure center is also appear to the negative force surface of vane on normalwise section. At t=7s (vane rotates 54°), strong low pressure vortex moves to downstream zones, strip low pressure zone appeared on the normalwise section. At t=9s (vane rotates 63°), the negative pressure center transfer to near the vane trailing edge, a zone of negative pressure on normalwise section decomposition for multiple small negative pressure center transfer to the downstream region. At t=11s (vane rotates 72°), a large area of low pressure is reformed near the negative pressure center of the guide vane, which influence the flow field in the downstream regions. At t=13s (vane rotates 81°), the downstream low pressure zones separate into a large range of low pressure zones and a strip of low pressure near negative force surface of vane. At t=15s (vane rotates 90°), the low pressure zone continues to develop, in the distance by three times of chord lengths position dissociate a negative pressure center. The negative pressure center range gradually expanded on normalwise section, the obvious oscillation flow around vane is formed.
3.3.2. The changes of wake structure in the process of moving vane. In order to analyze 3D topological structure of the wake, including temporal and spatial distribution characteristics, the non-dimensional $\lambda_2$-criterion was used to identify the wake. The wake distribution at typical moment of moving vane was shown in Figure 6. As indicated, with the moving vane the development of wake and vortex structures in flow field demonstrated obvious strong three-dimensional, nonlinear and asymmetric distribution characteristics. The evolution of wake is from the order to the disorder. At $t=1s$ (vane rotates 9°), with the vane begin moving the large scale of "starting vortex" was reeled up in the flow field, the near wake is formed by the trailing vortex and vortex shedding. At $t=3s$ (vane rotates 27°), on both sides of vane have many large scale vortex structure one after another to roll up, continuously to promote the downstream direction, in the development process of the vortex size increases, and the distance between the vortex structure also constantly stretched. At $t=5s$ (vane rotates 45°), large eddies break into small eddies in downstream development zone, no longer appearing to the same magnitude size of vortex structure. At $t=7s$ (vane rotates 54°), the continuous roll up and development mechanism of vortex reflect the space quasi periodic of the coherent structure. At $t=9s$ (vane rotates 63°), the shear layer separation and roll up vortex in the wake around the spiral spread out formation to the complex three-dimensional structure. At $t=11s$ (vane rotates 72°), the energy gradually transmit from large scale eddy to small scale eddy in downstream propagation of shedding vortex, because of fluid viscosity, until a certain level of small scale eddies to transfer energy by viscous dissipation, and shedding process is caused by the asymmetry of the vortex development. At $t=13s$ (vane rotates 81°), vortex topology like a long rope appear in the region near the outlet, a braided vortices is formed. At $t=15s$ (vane rotates 90°), through the interaction between the vortex structure, the energy transfer process of different scale eddy is completed. The energy dissipation and transfer part always occur in the place of vortex violently. In the guide vane turn off process, various scales of vortex structures appear, with the nonlinear effect of the increase in different scales of the eddies in the evolution process between continuously exchange energy, induce the formation of complex nonlinear large eddy coherent structure and unsteady dynamic interactions.

In summary, in the guide vane closing process, the downstream area of vane continued to appear the phenomenon of alternating vortex structures of different scales. This is due to the different scales of the vortex structure in the evolution process because of the energy transfer mode alternation, thus
inducing a flow around unsteady flow oscillation phenomenon.

![Figure 6. Wake structure distributions flow around kinetic guide vane.](image)

(a) t=1s, (b) t=3s, (c) t=5s, (d) t=7s, (e) t=9s, (f) t=11s, (g) t=13s and (h) t=15s.

In 8 core CPU, 24G memory workstations, we simulate all the numerical results produced animation display, realistically reproduce the large eddy rolled up, vortex merger and multiple large eddy interaction process.

4. Conclusions
The numerical results indicated that, after full development zone, the rich coherent vortex and interaction in flow around the vane were improved, and the distribution characteristics of temporal evolution wake was obtained. The immersed boundary method can better simulate complex shape boundary surface of structure and deal with flow around moving boundary problems, and can
accurately calculate the wake flow around airfoil vane. The immersed boundary method by adding the corresponding source terms represents fluid-structure interaction in the Navier-Stokes equations. The biggest advantage of this method lies in that the Cartesian grid can achieve almost any complex geometry boundary, simple and efficient without loss of accuracy. In addition, the immersed boundary method can be easily combined with advanced turbulence model (LES and DES), and wake structure to simulate the fine, reasonable and intuitive. The calculation is also superior.

The objective of the calculation is to reveal the guide vane dynamic flow around oscillating wake physical and mechanical mechanism. In order to make the process of the evolution of the wake structure not influenced by the channel geometries, and to reveal the internal mechanism of oscillating wake, the vane passage is simplified to the channel. Based on the simulation results of the vane-channel model, despite the large differences between the channel geometry and the true vane passage, the flow characteristics of flow around a vane in the near wall region of the guide vane are similar to that of the real vane passage. Therefore, the results based on the flow characteristics obtained by the vane-channel model can be used to illustrate the dynamic flow around the real vane passage.

Acknowledgments
The project is financially supported by National Natural Science Foundation of China (NSFC) (Grant No. 51541913 and 51279071) and Yunnan Agricultural University Natural Science Foundation for Young Scientists (No. 2015ZR17).

References
[1] Aparna P, Pothanna N, Ramana Murthy J V and Sreelatha K 2017 Alexandria Eng. J. 56 679-85
[2] Aparna P, Pothanna N and Ramana Murthy J V 2019 Numerical Heat Transfer and Fluid Flow (Singapore: Springer) pp 285-93
[3] Aparna P and Ramana Murthy J V 2007 Bull. Calcutta. Math. S. 99 293-304
[4] Hay A, Etienne S, Garon A and Pelletier D 2015 Comput. Methods Appl. Mech. Eng. 295 172-95
[5] Peskin C S 1972 J. Comput. Phys. 10 252-71
[6] Gronskisa A and Artanab G A 2016 Comput. Fluids 124 86-104
[7] Chiu P H, Lin R K and Tony W H S 2010 J. Comput. Phys. 229 4476-500
[8] Yang F C and Chen X P 2015 J. Hydrodynam. 27 658-67
[9] Liao C C and Lin C A 2012 Comput. Methods Appl Mech. Eng. 213-216 58-70
[10] Mori Y and Peskin C S 2008 Comput. Methods Appl Mech. Eng. 197 2049-67
[11] Zhang H, Yu A B, Zhong W Q and Tan Y Q 2015 Int. J. Heat Mass Transf. 91 178-89
[12] Ilinca F and Hétu J F 2010 Comput. Fluids 39 1656-71
[13] Seo J H and Mittal R 2011 J. Comput. Phys. 230 1000-19
[14] Nicolau L, Jung S Y and Zaki T A 2015 Comput. Fluids 119 101-14
[15] Sui Y, Chew Y T, Roy P and Low H T 2007 Int. J. Numer. Methods Fluids 53 1727-54
[16] Yang J, Preidikman S and Balaras E 2008 J. Fluids Struct. 24 167-82
[17] Sun X S and Sakai M 2016 Chem. Eng. Sci. 139 221-40
[18] Ji C, Munjiza A and Williams J J R 2012 J. Comput. Phys. 231 1797-821
[19] Ilinca F and Hétu J F 2012 Comput. Fluids 59 31-43
[20] Zhang H, Trias F X, Oliva A, Yang D M, Tan Y Q and Shu S 2015 Powder Technol. 272 1-13
[21] Weymouth G D and Yue D K P 2011 J. Comput. Phys. 230 6233-47
[22] Khosronejad A, Kang S, Borazjani I and Sotiropoulos F 2011 Adv. Water Resour. 34 829-43
[23] Anderson D M, McFadden G B and Wheeler A A 1998 Annu. Rev. Fluid Mech. 30 139-65
[24] Scardovelli R and Zaleski S 1999 Annu. Rev. Fluid Mech. 31 567-603
[25] Mittal R and Iaccarino G 2005 Annu. Rev. Fluid Mech. 37 239-61