The Optimal Design based on CFD combined with CAD for Turbine Runner

Liying Wang
College of Water Conservancy and Hydropower, Hebei University of Engineering, Handan 056021, China
Email: 2000wangly@163.com

***Abstract*—Modern design method can not only make up the shortcomings of the traditional design method, but also be compared on the computer, it can save a lot of trial and testing costs in new product development and play an important role in optimizing the existing products. This paper studies the modern design methods of hydraulic machinery by the use of computational fluid dynamics (CFD) combined with CAD, which improves the blade shape according the analysis results to achieve the best design. Firstly the CFD calculation of the flow is applied to the runner blade of francis, then we analysis and evaluate the flow character and cavitation characteristics of the flow field in the inlet of the blade under the large flow, finally we carry on improving the inlet flow and cavitation characteristics by rebuilding the blade shape, the simuliaton results show the modified runner has better resistance to cavitation**

***Index Terms*—Francis turbines, computational fluid dynamics, CAD, modern design method**

I. INTRODUCTION

The increasing efficiency enhancement of hydraulic machinery and cavitation performance, has always been an important part of the area of hydraulic machinery[1], the particular study of its working mechanism, the exploration of advanced design methods, and the improvement of product design and manufacturing level are necessary to obtain high quality products [2, 3]. Therefore, along with the rapid development of computer and numerical simulation technology, many researchers pursue a study on modern design of hydraulic machinery using advanced tools and methods throught analysizing internal flow field of hydraulic machinery, they strive to fundamentally solve the problems in the presence of hydraulic machinery which were deal with relying on the previous experience and approximation method with the uncertainty of results and the high cost.

The computational fluid dynamics based on the classical fluid dynamics and numerical calculation method is a new independent discipline [4, 5], through the numerical computation of computer and images show, it quantitatively descripts numerical solution of the flow field in time and space, so as to achieve the purpose of the research of physical problems [6, 7]. It has the characteristics of theory and practice which has established many involved theories and methods, which has provide an effective computing technology for modern science for many complex flow and heat transfer problems, the application of CFD is closely related to the development of computer technology, and the CFD software has become a powerful tool which is used to solve all kinds of fluid flow and heat transfer problems, it has many successful applications for various kinds of technical fields of science including water conservancy, shipping, marine, environment, food, fluid machinery and fluid engineering, etc [8, 9, 10].

At present, in the design process of hydraulic machinery, CAD and CFD have gradually become important means to design and optimize many flow components; they can greatly reduce the experiment times of hydraulic machinery design, save cost, and shorten the development cycle. Now, in actual application, runner blades of francis turbines used usually adopts the traditional two-dimensional flow method, after the actual test, there unavoidably exists some defects which are not discovered and avoided throught the traditional methods. So, this paper calculates internal flow field of a number of conditions of francis turbines using CFD technology in detail, and we analyze flow character and cavitation characteristics of the runner and improvement measurement of runner cavitation performance, finally the improvement scheme is put forward.

II. THE MODERN DESIGN METHOD BASED ON CFD COMBINED WITH CAD

CFD technology is one of the key technologies which adopts the modern design method to hydraulic turbine, at present, CFD technology is considered as the main assistant method in a lot of design and modification [4, 10]. Under the influence of the technology, it gradually changes from the traditional design method to the modern design method based on the CFD technology [11]. CAD-CFD modem design method is based on the theory of CAD and CFD, CAD technology is used to fix port shape, CFD technology is used to analyse the internal three-dimensional flow field which includes the characteristics of flow pattern, the flow field distribution and the

Manuscript received Sept. 10, 2011; revised Oct. 17, 2011; accepted Oct. 28, 2011.
project number: 2011138 and E2010001026

doi:10.4304/jsw.7.8.1721-1726

calculation of performance parameters, the goal of this method is to improve the performance index of turbine, judge the rationality of the design, and then optimize the geometry size. Through the optimization of the related geometrical parameters, we can ensure that the turbine has better comprehensive performances. The design process based on CFD combined with CAD for turbine is shown in Fig. 1.

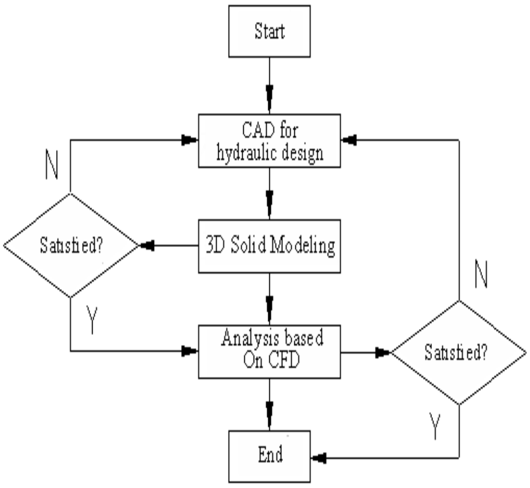


Figure 1. The design process based on CFD combined with CAD for turbine

III. THE GEOMETRICAL MODEL OF TURBINE RUNNER

In order to accurately simulate the inner flow of turbine runner, the runner and three-dimensional flow calculation domain firstly should accurately carry on geometry modeling which is the necessary shape boundary condition. In the analysis of CFD, it is required to provide the accurate numerical value data of the blade surface. The geometric shape of turbine flow surface is more complex, in the design process; curve or surface will be used in every step, the transformation calculation from curve to curve and from curve to surface is throughout the design process. Therefore, the proper mathematical method should be choosed to improve the interaction design ability of CAD system, so as to adapt to design personnel who carry on the design with experience.

Non-uniform rational b-spline (NURBS) method has the strengths to describe the shape freely and has the advantages to precisely show the elementary curve of surface. A uniform mathematical model is used to realize all sorts of curves of the surface in the method of NURBS, at the same time, the control factor and its power right are used to provide sufficient flexibility. NURBS method is applied to turbine design; it can not only raise the interactivity of flexibility of geometry model, but only realize the numericalization relying on less data (the control points and its power factor).

These limited data are used to compute the control points and the corresponding power factor of the NURBS curve and surface which are used to realize accurate description of any point coordinate at the surface. The practice shows that, as long as the control points and its power factor can be reasonably used, we can construct a compatible part surface with the real one.

Referring the model diagram of the turbine blade [12], the high section contour line in the working and back of the blade can be obtained, which are shown in Fig. 2 and Fig. 3. The inverse calculation method of bi-cubic b-spline curve is used to separately calculate the working surface and the control points of back face; the calculation procedure is as follows:

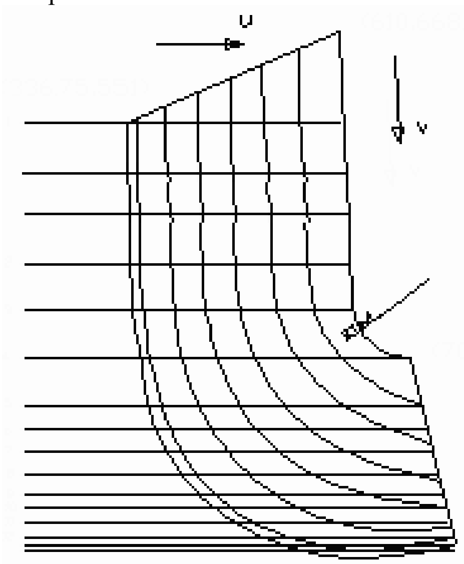


Figure 2. The axis pojection of turbine blade

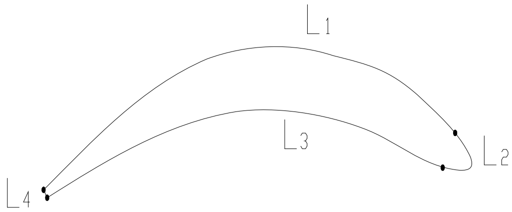


Figure 3. The contour line of horizontal cross section of blade

(1) The number of level section contour lines is $m+1$, each level section online has $n+1$ points, the spatial point coordinates of the corresponding blade surface is $p_{i,j}(x_{ij},y_{ij},z_{ij})$, $(i=0,1,2,\ldots,m, j=0,1,2,\ldots,n)$.

(2) The crown cut lines and level section lines are used as the surface cut lines of the blade surface in u direction, according to (1) and cubic B-spline inverse calculation method, the control points $c_{r,j}$ of various surface cut lines are calculated.

$$\sum_{r=0}^{m+2} c_{r,j} N_{r,3}(u_{3+i}) = p_{i,j} \qquad (1)$$

(3) the control points $c_{r,j}$ $c_{r,\,j}$ (r= 0,1,2,..., m+2; j= 0,1,2,...,n) is used as the interpolation points, in v direction, according to the type (2) and cubic b-spline inverse calculation method, the control points $d_{r,s}(r=0,1,2,\ldots,m+2;\ s=0,1,2,\ldots,n+2)$ of various surface cut line are calculated.

$$\sum_{s=0}^{n+2} d_{r,s} \cdot N_{s,3}(v_{3+j}) = c_{r,j} \tag{2}$$

Then, set each point right factor $\omega_{r,s}$=1.0, bi-cubic NURBS surface is described by using the followed equation:

$$p(u,v) = \frac{\sum_{r=0}^{m+2}\sum_{s=0}^{n+2} \omega_{r,s} d_{r,s} N_{r,3}(u) N_{s,3}(v)}{\sum_{r=0}^{m+2}\sum_{s=0}^{n+2} \omega_{r,s} N_{r,3}(u) N_{s,3}(v)} \tag{3}$$

Where, $N_{r,3}(u)$ and $N_{s,3}(v)$ are cubic B-spline basis functions.

According to (3), the blade surface can be generated, if the blade surface is more complex, we can generate subdivision, and then connect them together. As is shown in Fig. 4, according to the characteristics of the blade, we will take it into three parts, the first part is from the crown to level section 3, the second part is from the level section 3 to 4, the third part is from the level section 4 to 13. In the second part, in order to fit better the inlet side of the transition, more than three level cut lines should be add between section 3 and 4 based on the design information in level section.

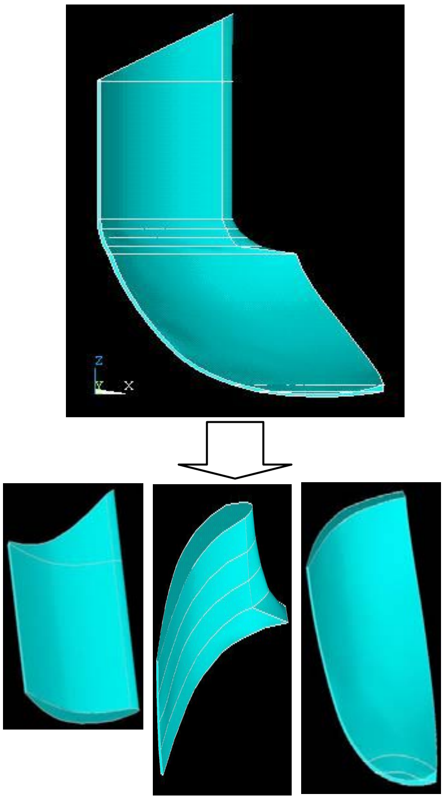


Figure 4. The blade model of schematic diagram

When the third part is finished, a small part between the level section 13 and the bottom also should be made. The above blade geometry model is used to further establish wheel entity which is shown in Fig. 5; it can realize the geometrical modeling of the runner.

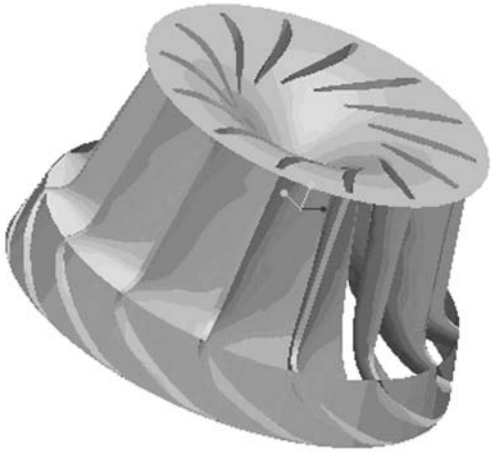


Figure 5. The solid diagram of francis turbine runner

IV. The CFD Analysis for Inner Flow of Runner

A. Grid Division

The blade intermediate stream surface is used as a interface surface, we extract flow field of a blade and the extension to the tail pipe outspread part of the inlet and outlet of guide leaves which can constitute a calculation field, then grid calculation is divided into 520000 grids. The calculation grid of three-dimensional computational domain is illustrated in Fig. 6.

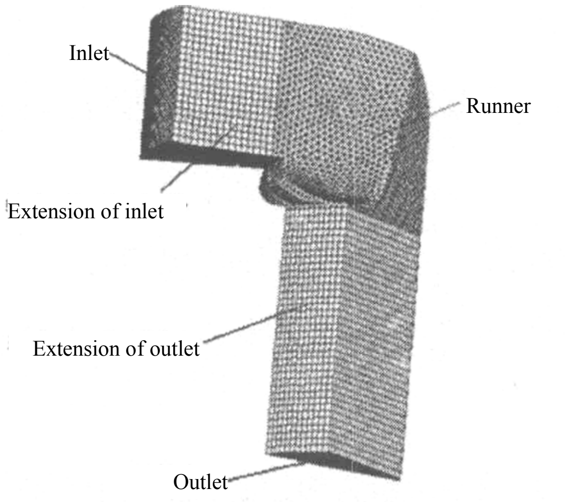


Figure 6. The calculation grid of three-dimensional computational domain

B. Boundary Conditions

(1) Inlet boundary conditions

According to the inlet flow of guide vane, each velocity component in all directions at the inlet of the calculation field is given: $u = u_m, v = v_m, \omega = \omega_m$, the agitation kinetic energy k and its dissipation rate ε respectively are obtained by the following empirical formula:

$$k_{in} = 0.005(u_{in}^2 + v_{in}^2 + \omega_{in}^2),$$
$$\varepsilon_{in} = 2C_\mu \cdot k_{in}^{3/2} / D. \quad (4)$$

where D is inlet equivalent diameter.

(2) Outlet boundary conditions

The runner outlet is considered as the pressure outlet whose relative pressure is set at 0.

(3) Wall boundary conditions

In the rotation area, we use the sliding surface without conditions; the runner together with the wall surface is set as relatively static, and the near wall region adopts wall function method.

(4) Cyclical conditions

In the calculation field between the upstream of inlet and downstream of outlet, both sides are given to a periodic condition. Periodic boundary condition is the boundary condition of periodic rotation at both sides of flow channel.

$$\phi/R = \phi/L\,(\phi = \mu, v, \omega, p, k, \varepsilon) \quad (5)$$

Both sides of flow channel are used as boundary conditions of periodic rotation.

C. CFD Calculation

This paper selects three typical operation points with big flow as analytical operation points [13, 14]: Q_{11} is set to 1100 (m^3/s), and n_{11} is respectively set to 90, 75 and 60 (*RPM*), then the k-ε turbulence model is adopted for the calculation.

V. Flow Field Distribution of Runner

A. Velocity Distribution

We calculate the speed distributions corresponding to the three operation points, which are demonstrated in Fig. 7 to Fig. 9. It can be seen that from Fig. 9 that the part near to the lower flow surface is not reasonable, the inlet setting angle is too small which may enable the inlet to be out of flow, especially in some load operating condition the situation will probably be more serious, this will lead to larger impact loss of the inlet flow and bad cavitation characteristics.

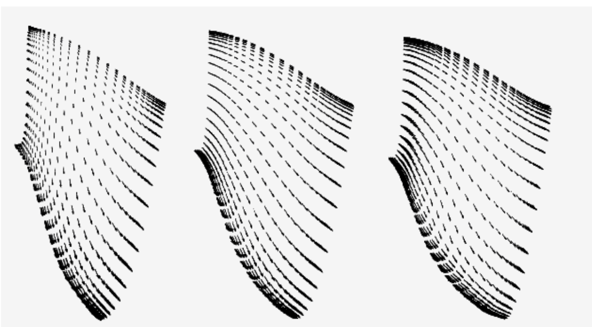


Figure 7. Near positive relative velocity vector diagram of 3 operation points

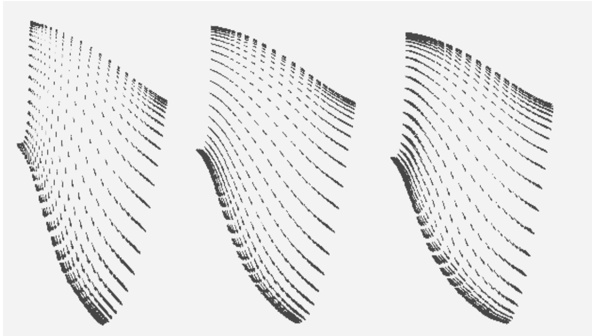


Figure 8. Near negative relative velocity vector diagram of 3 operation points

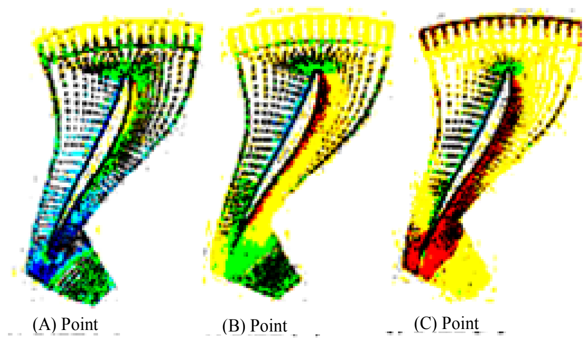
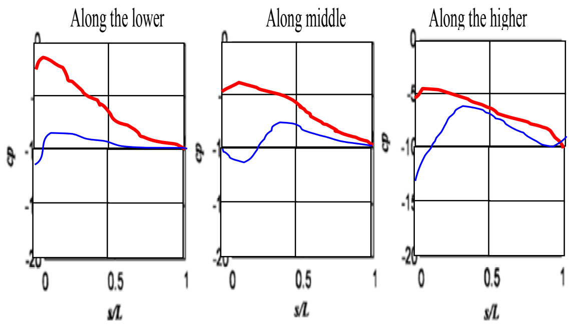


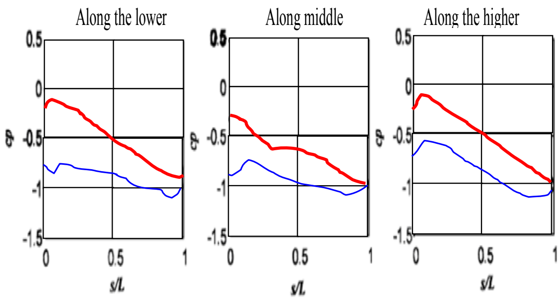
Figure 9. the relative velocity vector diagram near to the lower flow surface

B. Pressure Distribution

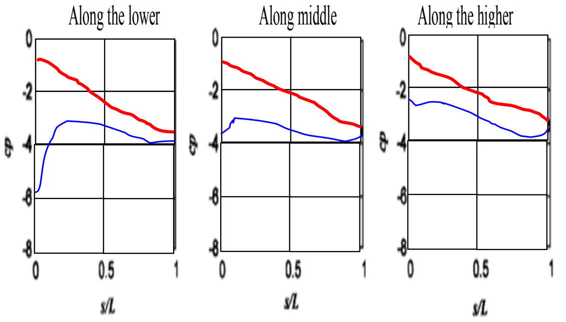
We calculate the pressure distribution of the three operating points, which is shown in Fig. 10.



(a) The first operating point



(b) The second operating point



(c) The third operating point

Figure 10. The distribution curve of pressure coefficient along the flow direction of blade surface

Fig. 10 depicts the pressure coefficient curves respectively corresponding to the positive and negative of blade along three different directions in different working conditions. The three operating points all have large pressure gradient in the back of blade head, the pressure of each working condition in the blade near to the back of the above inlet is slightly lower, except for the lowest pressure point in blade head. The pressure distribution in the crown of working conditions is more uniform and the pressure gradient is small, so we draw a conclusion that the blade back near to the lower ring is the cavitation area, it is in agreement with practice which is shown in Fig. 11.

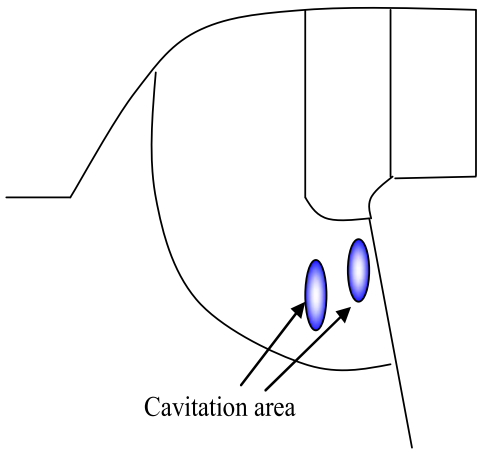


Figure 11. The schematic diagram of cavitation area of runner blade

According to the above analysis results of CFD, we carry on a shape modification for blade, the setting angle of the inlet is increased by 5º in Fig. 11; to relieve the flow separation of inlet back and reduce impact loss of inlet flow, the setting angle of the outlet is increased by 8º in Fig. 12. We make the leaf form near to lower ring flat in order that the streamline velocity moment variation ($\Delta Vu{\cdot}r$) close to the lower ring area reduces, it can improve the pressure of blade outlet near to lower ring.

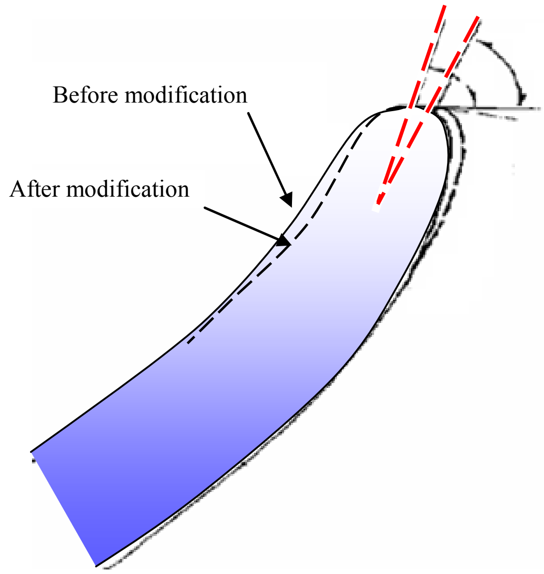


Figure 12. The blade modification diagram near to the lower ring

After carrying on a CFD analysis for the runner it can be seen that the pressure of blade outlet near to lower ring is improved, Fig. 13 shows the cavitation characteristics are enhanced after the modification.

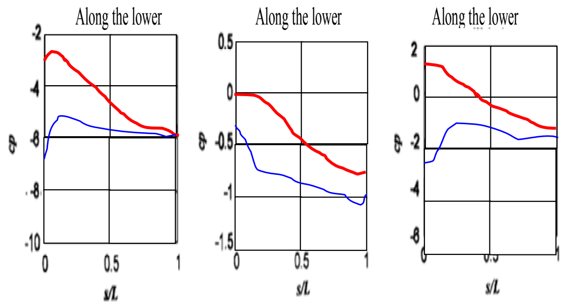


Figure 13. The distribution curve of pressure coefficient corresponding to the negative back of blade near to the lower ring after modification

VI. Conclusion

Advanced CAD-CFD modern design method, which is gradually becoming the main method for turbine design, is increasingly playing a larger role both in the design of new model and in the old model transformation. With the CAD technology, we can realize three-dimensional modeling of flow parts, relying on the CFD technology, we can realize the 3D flow numerical simulation and performance prediction, and then we can design turbines with a comprehensive performance through repeated modification. The article adopts CFD-CAD technology to the runner of francis turbine in order to improve the cavitation characteristics and the flow pattern of inlet. Firstly, the CFD calculation results are referred, and then we make a local modification for the geometry shape of runner blade near to the lower ring using the CFD technology, finally we carry on a numerical simulation analysis for the inner flow field of the modified runner, it is proved that the modified runner has better resistance to cavitation. Our future research is to improve the CFD method based on CAD and apply it for other the optimal design engineering problems.

ACKNOWLEDGEMENTS

This work is supported by the Science and Technology Research Project of University of Hebei Province No. 2011138, and the Natural Science Foundation of Hebei Province of China No. E2010001026.

REFERENCES

[1] Mural Yuichi, Yamamoto Fujio, “Quadrant analysis of bubble induced velocity fluctuations in a transitional boundary layer”, Journal of Hydrodynamics, Vol.21, No.1, 2009, pp91-96.

[2] YUAN Shou-qi, NI Yong-yan, PAN Zhong-yong, “Unsteady Turbulent Simulation and Pressure Fluctuation Analysis for Centrifugal Pumps”, Chinese journal of mechanical engineering, Vol.22, No.1, 2009, pp 64-69.

[3] HE cheng-lian, WANG zheng-wei, QIU hua, “The experimental study of internal pressure fluctuation of draft tube”, Journal of Machinery Industry, Vol.38, No.11, 2007, pp62-65.

[4] Wang Fujun, “CFD Software Principle and Application”, Tsinghua University press: 2004.

[5] Liu Houlin, Guan Xingfan, Shi Weidong, Yang Jingjiang, “The Characteristics and Development on Pump CAD Technology in the Domestic”, Fluid Machinery, Vol.30, No.3, pp26-29.

[6] RRN Jing, WU Yulin, YANG Jianming, ZHANG Wei, CAO Shuliang, “CFD analysis and optimum design of hydro turbine runner”, Journal of Engineering Thermophysics, Vol.21, No.3, 2000, pp316-320.

[7] JIAN hua, YAN Zhong-min. “Hydraulic characteristics of bottom underlay-type pier for water-wing control”, Journal of Hydrodynamics, Vol.20, No.6, 2008, pp65-73.

[8] WANG Xiao-ping, ZHEN Ling, HUANG Hai, CHENG Bo-chuan, ”Research on Pressure Fluctuations Time Serials by Deterministic Chaos Theory and Information En tropy”, Chinese Journal of Scientific Instrument, Vol.23, No.6, 2007, pp596-599..

[9] WU Dazhuan, WANG Leqin, BAO Zhiren, ” Numerical Simulation of Flow in Open-type Mixed Flow Pump and Options of the Discharge Structures”, CHINESE Jorunal of Mechanical Engineering, Vol.44,No.8, 2008, pp90-96.

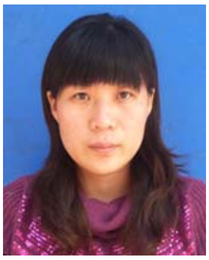
[10] MUGGLI F, HOLBEIN P, DUPONT P, “CFD calculation of a mixed flow pump characteristic from shut-off to maximum flow”, ASME Journal of Fluids Engineering, Vol.124, No.3, 2002, pp798-802.

[11] De Backer JW, Vos WG, Vinchurkar SC, Claes R, Drollmann A, Wulfrank D, Parizel PM, Germonpré P, De Backer W, “Validation of computational fluid dynamics in CT-based airway models with SPECT/CT”, Radiology, Vol.257, No.3, 2010, pp854-62..

[12] De Backer JW, Vos WG, Devolder A, Verhulst SL, Germonpré P, Wuyts FL, Parizel PM, “De Backer W,Computational fluid dynamics can detect changes in airway resistance in asthmatics after acute bronchodilation”, Journal of Biomechanics, Vol.41, No.1, 2008, pp106-113.

[13] TIAN Yabin, QI Xueyi, KONG yuan, HU Jiawang, “Three-dimensional modeling of blades of Francis turbines” Advances in Science and Technology of Water Resources, Vol.30, No.4, 2010, pp64-66.

[14] Jin Yong, “The Influence of Exit Structures on the Axial Distribution of Voidage in Fast Fluidized Bed”, China-Japan Symposium on Fluidization,1998,Vol.31, 1988, pp15-17



Liying Wang was born in Shijiazhuang, Hebei Province, China, in 1978. She received the B.S. and the M.S. degrees both in Institute of Mechanism, Shijiazhuang Tiedao University, in 2000 and 2003 respectively. Now she is Ph.D. candidate in Institute of Mechanical, Electronic and Control Engineering, Beijng Jiaotong University, her current research interests include intelligent computing, intelligent control theory, and control systems engineering.