ELSEVIER
Available online at www.sciencedirect.com
SciVerse ScienceDirect
Procedia Engineering 31 (2012) 187 – 195
Procedia Engineering
www.elsevier.com/locate/procedia

International Conference on Advances in Computational Modeling and Simulation

Optimum design on impeller blade of mixed-flow pump based on CFD

Jidong Li[a], Yongzhong Zeng[a*], Xiaobing Liu[a], Huiyan Wang[a]

[a]*School of Energy and Environment, Xihua University, Chengdu 610039, China*

Abstract

The three-dimensional flow field of the whole flow passage of a mixed-flow pump was numerically simulated by using CFD software on the basis of Spalart-Allmaras turbulent model according to the original design of the plant. Through analyzing the calculation results, the reason why the flow rate of this pump can not reach to the design requirements was found out. After replacing the impeller, a new pump impeller was optimally designed. The numerically simulation results show that the hydraulic performance of the newly designed impeller of the mixed-flow pump were obviously improved, and the engineering requirements of the owner were satisfied.

Keywords: Mixed-flow pump; Internal flow; Numerical Simulation; Optimal design

1. Introduction

Mixed-flow pumps play an extremely important role in the economic construction of China. They have been widely used in agricultural irrigation, flood control, urban water supply, the cooling water system of power plant, and other fields. The mixed-flow pump studied in this paper was used to take water from a river by a water company. During the installation and commissioning stage, technicians discovered that the flow rate of this pump was only 2950m^3/h when the head is 14m, which was far from meeting the design requirements of the owner (the flow rate shall be 5700m^3/h when the head is 14m). Entrusted by this company, optimal design of this pump was conducted with only replacing the impeller and without replacing the flow passage, fixed guide vane and motor.

*Corresponding author. Tel.: +86-28-88983769; fax: +86-28-87720521.
E-mail address:zyzzyzhome@163.com

1877-7058 © 2011 Published by Elsevier Ltd. Open access under CC BY-NC-ND license.
doi:10.1016/j.proeng.2012.01.1011

Impeller is an important flow passage component in a mixed-flow pump. To find out the causes of low hydraulic performance of the original pump, it is necessary to study the flow in the impeller. While the impeller of the pump is operating, the impeller will rotate, and the geometric shape of the flow passage will be distorted. The water flow in the impeller is a complicated three-dimensional turbulent flow. Therefore, observing the flow situations in the impeller through experiment not only will be a waste of time, but also will cause economic losses due to the shutdown. In recent years, with the rapid development of computer technologies and computational fluid dynamics, CFD software has been used to numerically simulate the three-dimensional viscosity flow in the impeller, and optimal design of the impeller on this basis has become a reality. By using CFD software and model test, YUAN Chuan-yuan [1] thought that the calculation results by numerically simulation software Fluent can truly reflect the flow in the impeller of a mixed flow pump on the premise that the turbulent model and boundary conditions are similar to the actual situations. He also conducted performance test to the designed mixed flow pump, and obtained the performance curve of the pump. The results showed that the numerically simulated results were in good coincidence with the experimental results, and the hydraulic model of the mixed flow pump designed according to the simulation results can meet the engineering requirements. MA Xi-jin and ZHANG Hua-chuan in [2] predicated the hydraulic performance of a three-level circulating mixed flow pump of a nuclear power station by CFD software. Meanwhile, they conducted performance test on clear water rig. The numerically simulation results were in good coincidence with the experimental results. The hydraulic performance of the pump was further improved by enlarging the section area of the volute and reducing the blade inlet angle of attack based on CFD analysis and the experimental results.

The entire flow field of this mixed-flow pump was numerically simulated by using Fine/Turbo8.7.2 CFD software. Without stopping the pump, the detailed flow parameters in the entire flow passage of the original pump were obtained. And then, the original impeller blade was optimally designed, and the entire flow field of the new pump was numerically simulated. The flow performance parameters and the details of internal flow field of this mixed-flow pump were given in diagrams. Main flow characteristics in the entire flow field of the original pump and the new pump under the design conditions were analyzed in detail. The causes of insufficient flow rate of the original pump were found out. The hydraulic performance of the pump was improved through reducing the number of blades and increasing the radial curvature of the original impeller blade. The technological transformation of the original pump was achieved.

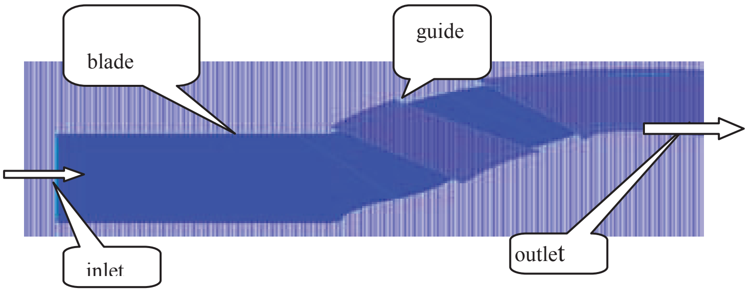
2. Analysis of three-dimensional modeling and numerical simulation of the whole flow passage of the original pump

2.1. Three-dimensional modeling of the flow passage and grid partition

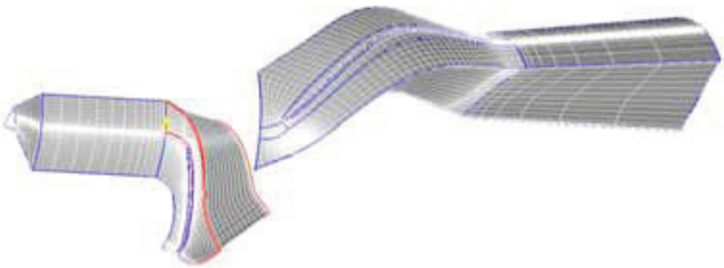
The three-dimensional model used in the calculation was a three-dimensional solid model made by 3D modeling software based on the design drawings provided by the plant. The main design parameters of the impeller of this mixed-flow pump were: 7 blades, rotational speed of 590r/min (rotational direction was negative direction of axis Z), inlet diameter of 600mm, and outlet diameter of 1000mm.

The whole flow passage of the entire pump includes inlet zone, impeller rotor zone, fixed guide vane zone, and outlet zone, as shown in Fig.1(a). The grids of impeller flow passage were partitioned by special rotating machinery grid generation module AutoGrid in NUMECA in [3]. The flow of this pump was axisymmetric. To reduce the computing workload, single blade flow passage was selected as the computational area with the total grid number of 1,991,688. During the grid generation process, the grid distance of the first layer near the wall of the impeller was selected as 0.013mm according to the selected

SA turbulent model. The value of Y^+ was controlled within the scope required by the low Reynolds number turbulent model. Grids near the wall (similar to boundary layer) were densified along the normal direction of the wall according to the law of geometric progression. Grids of single blade flow passage were shown in Fig.1(b). The three-dimensional solid model of the whole flow passage was shown in Fig.1(c).



(a) Axial side flow passage



(b) Grids of single bade flow passage

(c) Whole flow passage of the blade of mixed-flow pump

Fig.1. Three-dimensional modeling and grids of the flow passage of mixed-flow pump

2.2 Calculation method

The Euranus solver in NUMECA software package was used in the numerical calculation. This solver adopted the finite volume difference scheme of Jameson. Combined with Spalart-Allmaras turbulent model, the three-dimensional Reynolds average Navier-Stokes equations in the relative coordinate system were solved. Steady solution was obtained by using the explicit fourth order Runge-Kutta method time advance. To improve the calculation efficiency, multi-grid method, local time step, residual smoothing, and other methods were used to accelerate the convergence process in [4].

In the steady three-dimensional flow field calculation, the hydraulic performance parameters in different operating points under constant speed were obtained through continuously increasing the static pressure at the outlet boundary and changing the velocity at the inlet boundary. The performance curve of this pump was obtained in this way.

2.3 Boundary conditions

The boundary conditions: inlet flow rate and inflow angle were given for the inlet, the inlet direction of flow was the positive direction of axis Z. Average static pressure was given for the outlet. Adiabatic no gliding boundary conditions were adopted at the wall. The shroud connected with the impeller blade was static, and its clearance with the impeller blade was 0.5mm.

1) Impeller speed: 590r/min, rotational direction was the negative direction of axis Z.

2) Inlet conditions: axial inlet and given inflow angle. The atmospheric static pressure was 10,1325pa, and the inlet flow rate was gradually increased from 2000m^3/h to 6500 m^3/h.

3) Outlet static pressure: the outlet average static was gradually reduced according to the increasing of the inlet flow rate.

2.4 Convergence criteria

1) After dropping 5 magnitudes, the total residual will not drop and maintain slight oscillation.

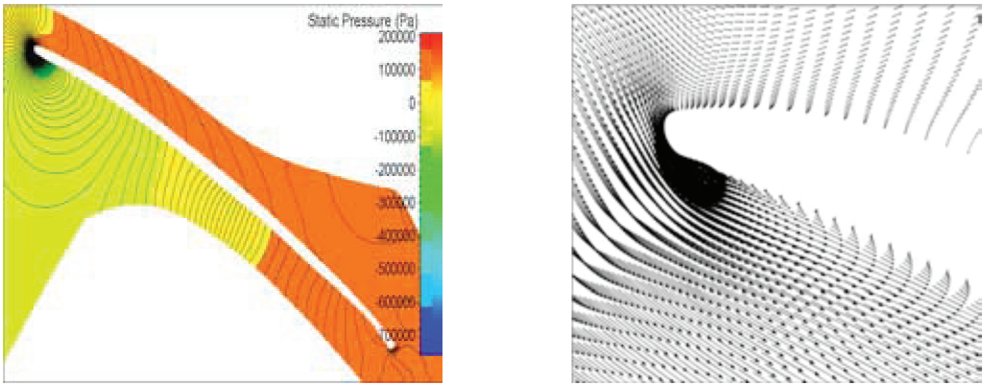
2) With the increasing of iteration steps, the general parameters (mass flow at the inlet and outlet, total pressure ratio, torque, power, etc.) will not change any more and will maintain that level.

2.5 Calculation results and analysis

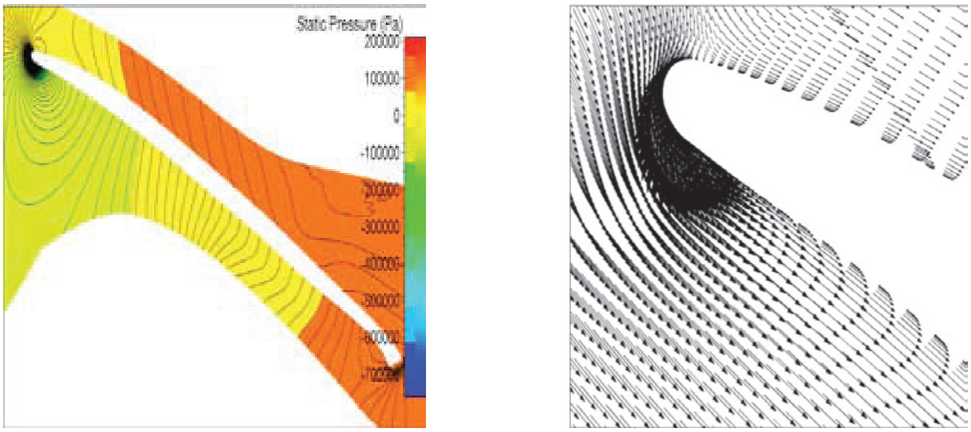
During the transformation process, the original flow passage and fixed guide vane were not changed, and only the impeller was changed. So only the pressure and velocity vector distribution diagrams of different blade height stream surfaces of the impeller blade in the actual operating points (head of 14m, flow rate of 2950m^3/h, and speed of 590r/min) were given in Fig.2 and Fig.3.

From the distribution of velocity vector at the inlet of the impeller blade in Fig.2, it can be preliminarily judged that the flow rate was bound to reduce under this operating condition due to the reducing of blade-to-blade flow area when vortexes appeared on the bade. That’s why this pump can not meet the actual requirements of the user. Seeing from the static pressure distribution of different blade height stream surfaces at the left of Fig.2, the static pressure on the operating surface of the blade are all above 100,000Pa, and the head can meet the requirements of the user. This is consistent with the actual operation results.

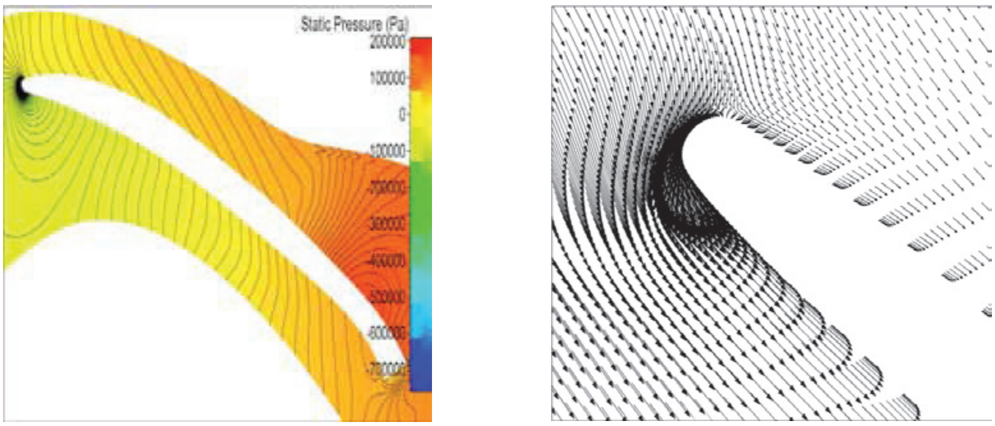
From Fig.3 the performance curve of this pump fitted according to the numerical simulation results, we can see that the hydraulic efficiency of the actual operating point of the original pump was only 58%, the flow fate was 2950m^3/s while the head was 14m, which was the low flow rate, low efficiency and high head operating zone.



(a) Static pressure distribution of 10% relative blade height stream surface and velocity vector distribution at the blade head



(b) Static pressure distribution of 50% relative blade height stream surface and velocity vector distribution at blade head 50%



(c) Static pressure distribution of 90% relative blade height stream surface and velocity vector distribution at blade head 90%

Fig.2 Static pressure distribution of the original blade at different blade height sections and the relative velocity vector distribution at the blade head

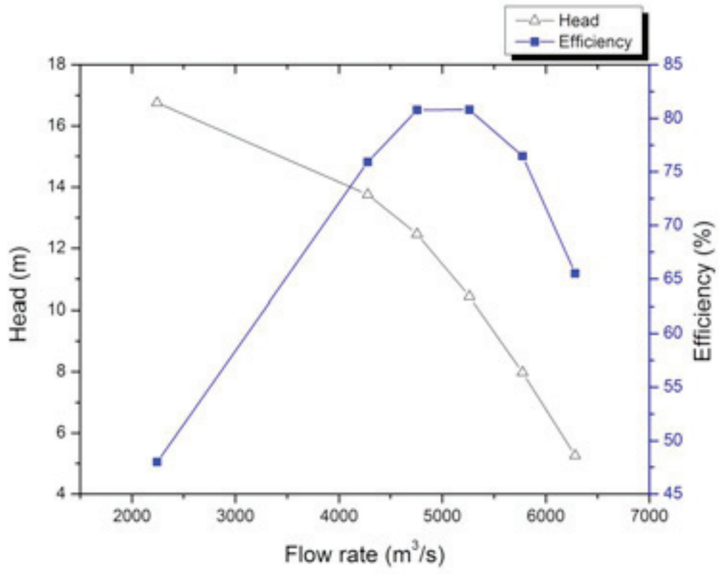


Fig.3 Fitted head-flow rate and efficiency-flow rate curve of various calculating points of the original pump

3. Optimal design of the pump impeller

To increase the flow rate of the pump without changing the head, the blades of the newly designed axial pump impeller were reduced from 7 to 5. By referencing the design parameters of excellent impeller blade with similar specific speed, the blade was designed into “C” type curved section to reduce the occurrence of vortexes at the head of the blade. The specific methods were to increase the blade inlet structure angle β_{1K} at the blade section as shown in Fig.4, and properly reduce the blade outlet structure angle β_{2K} to eliminate or weaken the affect of vortex, and increase the flow rate in [5].

3.1. Calculation methods and boundary conditions

The calculation methods were the same as the above original pump, and the boundary conditions were set as:

1) Impeller speed: 590r/min, rotational direction as negative direction of axis Z.

2) Inlet conditions: axial inlet and given inlet inflow angle. The atmospheric static pressure was 101,325pa, and the inlet flow rate was gradually increased from 1800m^3/h to 9000 m^3/h.

3) Outlet static pressure: the average static pressure at the outlet was gradually reduced with the increasing of inlet flow rate.

The static pressure distribution of different blade height stream surfaces and the relative velocity vector distribution at the blade head as shown in Fig.5 were calculated under the design conditions of the pump station (head of 14m and flow rate of 5700m^3/h). From the distribution of velocity vector at the inlet of the impeller blade in Fig.5, it can be preliminarily judged that the vortexes appeared at the blade inlet were obviously reduced. In this operation conditions, the flow rate was bound to increase. From

Fig.6 the performance curve of this pump fitted according to the numerical simulation results, we can see that its hydraulic efficiency was reached to 80.5% and the flow rate reached to 5700m^3/h while the head is 14m, which was operating in large flow rate, high efficiency and high head zone. The engineering requirements of the owner can be satisfied. This indicates that the optimal design scheme of this mixed-flow pump was successful.

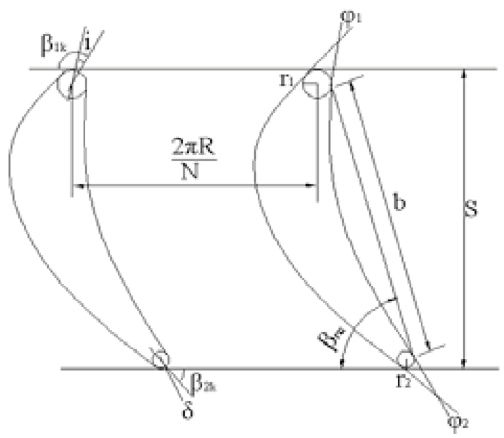
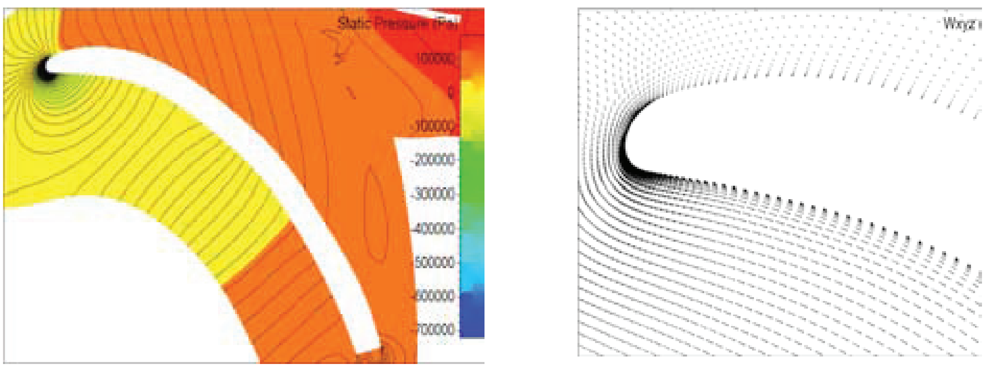
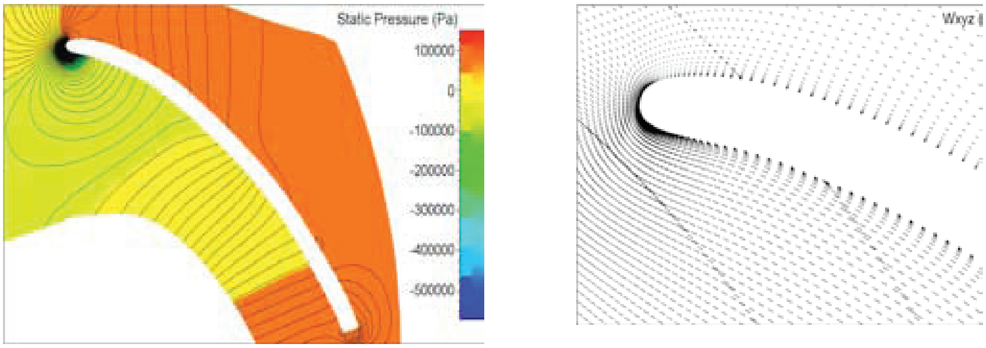


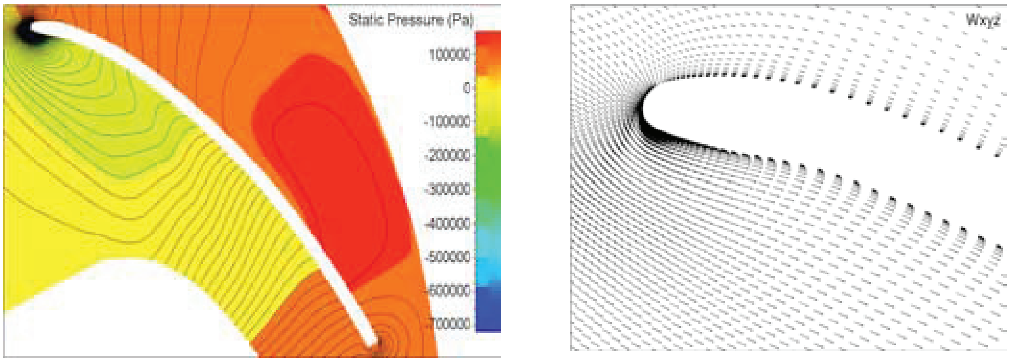
Fig.4 Schematic diagram of geometric parameters of the blade section



(a) Static pressure distribution of 10% relative blade height stream surface and velocity vector distribution at the blade head



(b) Static pressure distribution of 50% relative blade height stream surface and velocity vector distribution at the blade head 50%



(c) Static pressure distribution of 90% relative blade height stream surface and velocity vector distribution at the blade head

Fig.5 Static pressure distribution of the new blade at different blade height sections and the relative velocity vector distribution at the blade head

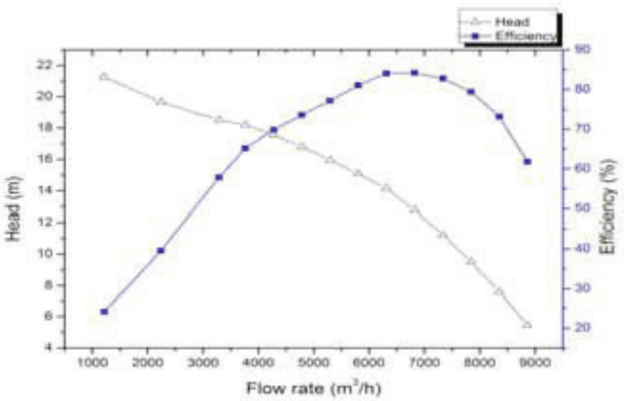


Fig.6 Fitted head-flow rate and efficiency-flow rate curve of various calculating points of the new pump

4. Conclusions

The whole flow passage of a mixed-flow pump was numerically simulated by using the FINETM/Turbo module of NUMECA software without stopping the pump. The performance curve of this pump and the details of internal flow under the actual operating conditions were obtained. The reason of not able to reach to the design requirements of the original pump under the actual operating conditions was found out. The flow rate of this pump was increased by reducing the number of blades, properly increasing the blade inlet structure angle ρ1K, reducing the blade outlet structure angle ρ2K, and other methods. Both the numerically simulation results and the actual operation conditions prove that the transformation of the original pump was successful.

5. Acknowledgements

This work was supported by the Open Research Funded Project of Provincial University Key Laboratory of Fluid Machinery in Xihua University (Grant No. SZJJ2009-012), and supported by the Innovation Fund of Postgraduate, XiHua, University (Grant No. Ycjj201129) and partially supported by Scientific Research Fund of SiChuan Provincial Education Department (Grant No. 11204026), meanwhile, this work was finished in the Provincial University Key Laboratory of Fluid Machinery in Xihua University. Jidong Li is a graduate student of 2009 in School of Energy and Environment, Xihua University.

References

[1] Yun Chuan-yuan. Numerical Calculation of Turbulent Flow and Performance Experiment of Mixed-flow Pump Impeller. Transactions of the Chinese Society for Agricultural Machinery. V01.39, No.3 2008. 52-55

[2] MA Xi-jin, ZHANG Huachuan, ZHANG Kewei. Numerical Simulation and Experiment Analysis of Thirdly Circulating Feed-water Mixed-flow Pump in Nuclear Power Station. FLUID MACHINERY. Vol.37, No.09, 2009 6-9

[3] AutoGrid™ V.5 user guide. Numeca; 2007.

[4] Fine™/Turbo V.8 user guide. Numeca; 2007.

[5] JIA Rui-xuan, XU Hong. Optimal design of low specific speed mixed-flow pump impeller. Journal of Drainage and Irrigation Machinery Engineering. Vol. No.02, 2010 98-102