Numerical Simulation of Different Turbulence Models in a Compact Return Diffuser

Ling Zhou, National Research Center of Pumps, Jiangsu University, China
Weidong Shi, National Research Center of Pumps, Jiangsu University, China
Ramesh Agarwal, Department of Mechanical Engineering & Materials Science, Washington University in St. Louis, Missouri, United States

ISROMAC
Since 1985
Long Abstract

Introduction

Pump is one of the most energy consuming devices widely used with general machinery in industrial applications. The information about the pump efficiency, manufacturing cost and reliability is required by the end-users before installation decisions are made. Pump manufacturing industry is currently focused on improving the energy efficiency of pumps while addressing stricter and stricter manufacturing constraints [1, 2]. This is particularly of interest in deep-well multistage pump design since it has high head requirement and severe constraints on operating dimensions. In our previous work [3, 4] we proposed a new type of deep-well centrifugal pump with a three-dimensional surface return diffuser, which has a compact size and high pressure conversion capability. However, additional research is needed to expand its application range and to further optimize its hydraulic performance.

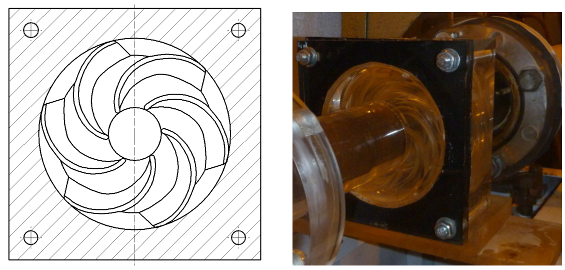
Due to the complex three-dimensional geometries of the impellers and diffusers, their design is a challenging task. Small changes in the design can result in significant changes in the internal flow structures affecting the hydraulic performance [5]. Over the years, considerable effort has been devoted towards the study of flow in impellers and diffusers using both the experimental and numerical methods [5-9]; the methods include the Particle Image Velocimetry (PIV), Laser Doppler Velocimetry (LDV), Computational Fluid Dynamics (CFD) and theoretical analysis. For example, Visser et al. [10] employed the LDV to measure the fluid flow in a low specific speed centrifugal impeller passage and showed that the core of the blade passage flow field could be described quite well by the two-dimensional potential theory at design discharge rates. Sinha and Katz [11, 12] used PIV to identify the unsteady flow structures and turbulence in a transparent centrifugal pump with a vane diffuser. Wuibaut et al. [13] measured the flow velocities in the impeller and vane-less diffuser of a radial flow pump with 2D PIV. Pedersen et al. [14] used both PIV and LDV to illustrate and measure the stall characteristics in the impeller at off-design conditions; both measurement techniques produced data in close agreement. Wu [15] used PIV to investigate the instantaneous internal flow field in a centrifugal pump with a volute. Feng et al. [16-18] employed PIV and LDV to study the unsteady flow in a radial diffuser pump and compared the measurements with CFD results using a variety of turbulence models. These and other investigations have significantly contributed to the understanding of the complex three dimensional flow fields in pumps. However, majority of research have focused on the rotating impellers. It appears from the literature that enough effort has not been devoted to understand the flow mechanisms inside the diffuser; it is especially true for the compact return diffusers.

Computational fluid dynamics (CFD) have been became the main method to study the pump inner flow features. Detached-eddy simulation (DES) is convincingly more capable presently than either unsteady Reynolds-averaged Navier-Stokes (RANS) or large-eddy simulation (LES) [19]. RANS models is the foundation of CFD, which have been wildly used to predict the pump performance and investigate the flow patterns in pump or turbo machinery area, a lots of good agreements by comparing with experiments results have been presented by many authors. RANS models can be adjusted to predict boundary layers and their separation well, but not large separation regions. DES models have been specifically designed to address high Reynolds number wall bounded flows, where the cost of a near-wall resolving LES would be prohibitive. It is important to understand the difference and features of different turbulence models. In the present work, both CFD simulations and two dimensional PIV measurements have been conducted to investigate the internal flow field in a compact return diffuser. The phase-averaged velocity field obtained by 2D PIV measurements is compared with the CFD results using different turbulence models.

1.Methods

1.1. Pump Geometry and Test rig

A single-stage pump section is chosen from a multistage deep-well centrifugal pump, the detailed geometry parameters and characteristics of the model pump are given in our previous paper [3, 20, 21]. In order to reduce the refraction in the PIV measurements, the outside wall is extended to a square type wall. The diffuser and the entire outside wall are manufactured with transparent plexiglas as shown in Fig.1 (b).



(a) Axial drawing (b) Diffuser model in the test rig

Figure 1. Diffuser Geometry and Solid Model

Figs.2 and 3 show the schematic of the test rig and the PIV system, which is also includes the single-stage pump cross-section. The PIV experiment system consists of a laser, a CCD camera, a synchronizer, a photoelectric encoder and a data processing computer. A variable frequency control cabinet is used to adjust the motor rotating speed to 2850r/min (pump rated speed). A plane mirror with central holes is used to refract light; it is placed at 45 deg to the axis of the test rig. The tracing particles used are Al_2O_3 powder of approximately 1μm diameter, which flows with the liquid synchronously and scatter the laser effectively.

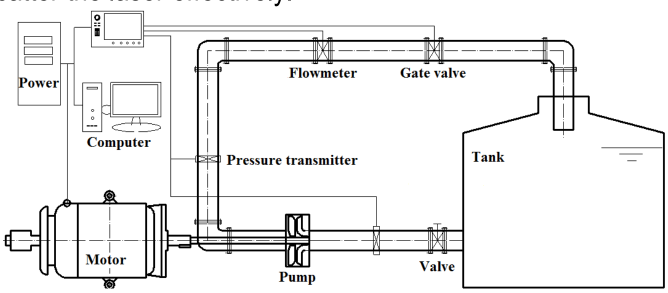


Figure 2. Test Rig Schematic

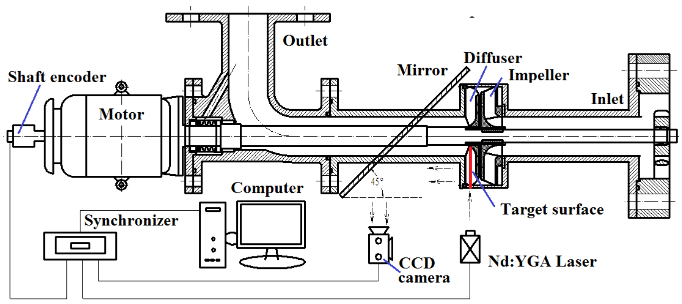


Figure 3. Schematic of Test Rig and PIV System

1.2. Numerical Setup

Transient three-dimensional incompressible simulations are performed using ANSYS-Fluent 14.5 code. The time step in the transient simulation is 5.848×10^{-5} s, corresponding to 1° of the impeller rotating speed, so one complete revolution is performed each 360 time steps. Numerical convergence is set to a maximum of 10^{-5} and periodicity solution is fulfilled after seven impeller revolutions. Before the transient simulation, a steady simulation was carried out by using Frozen Rotor. When achieving the setting convergence, the steady simulation was used as an initial value to start the transient sliding mesh simulation. Sliding mesh model is a special case of general dynamic mesh motion wherein the nodes move rigidly in a given dynamic mesh zone.

The computational domain was created based on the actual pump dimensions in PIV systems, as shown in Fig.4. After modeling in Pro/E, the geometric model was imported to ANSYS-ICEM for further processing. The entire computational domain was meshed with the high-quality structured grid based on the Q-type and Y-type block topology. Grid sensitivity analyses for a similar case have been reported in our previous work [21, 22]. Fig.5 shows a 3D view of the mesh in the impeller and the diffuser. To be special, the value of y^+ was less than 3 in the entire computational domain for DES and SST k-ω model with low-Re corrections. For SST k-ω model, the boundary mesh was reset to make sure the y^+ is around 30. About the boundary conditions, the mass flow inlet associated with the initial gauge pressure is set at the pump inlet. Pressure outlet boundary conditions are used at the pump exit. No-slip wall conditions are used at all the physical surfaces.

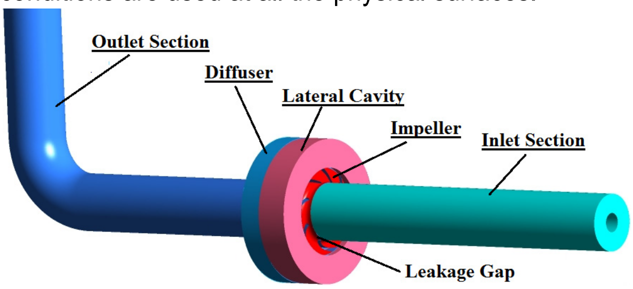
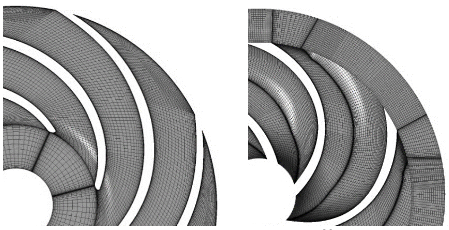


Figure 4. Computational Domain



(a) Impeller (b) Diffuser

Figure 5. Sketch of the Structured Grid

References

[1] Gopalakrishnan, S., 1999, “Pump research and development: past, present, and future - an American perspective,” *J. Fluids Eng. - Trans ASME*, Vol. 121, pp.237-247.

[2] Hergt, P. H., 1999, “Pump research and development: past, present, and future,” J*. Fluids Eng. - Trans ASME*, Vol. 121, pp.248-253.

[3] Zhou, L., Shi, W. D., Lu, W. G., et al., “Numerical investigations and performance experiments of a deep-well centrifugal pump with different diffusers. *J. Fluids Eng. - Trans ASME*, 2012, Vol. 134, pp.0711002.

[4] Shi, W. D., Lu, W. G., Wang, H. L., and Li, Q. F., 2009, “Research on the theory and design methods of the new type submersible pump for deep well,” *FEDSM 2009*, Vol. 1 (Pts A-C), pp. 91-97.

[5] Westra, R. W., Broersma, L., van Andel, K., and Kruyt, N. P., 2010, “PIV measurements and CFD computations of secondary flow in a centrifugal pump impeller,” *J. Fluids Eng. - Trans ASME*, Vol. 132(6), pp.061104.

[6] Akin, O., and Rockwell, D., 1994, "Flow structure in a radial flow pumping system using high-image-density Particle Image Velocimetry," *J. Fluids Eng. - Trans ASME*, Vol. 116, pp. 538-544.

[7] Myers, K. J., Ward, R. W., and Bakker, A., 1997, "A digital Particle Image Velocimetry investigation of flow field instabilities of axial-flow impellers," *J. Fluids Eng. - Trans ASME*, Vol. 119, pp. 623-632.

[8] Eisele, K., Zhang, Z., and Casey, M. V., 1997, "Flow analysis in a pump diffuser—Part 1: LDA and PTV Measurements of the Unsteady Flow," *J. Turbomach. - Trans ASME*, Vol. 119, pp. 968-977.

[9] Wernet, M. P., 2000, "Development of digital Particle Imaging Velocimetry for use in turbomachinery", *Exp. Fluids*, Vol. 28, pp.97-115.

[10] Visser, F. C., Brouwers, J. J. H., and Jonker, J. B., 1999, “Fluid flow in a rotating low-specific-speed centrifugal impeller passage,” *Fluid Dyn. Res.*, Vol. 24, pp. 275-292.

[11] Sinha, M., and Katz, J., 2000, "Quantitative visualization of the flow in a centrifugal pump with diffuser vanes. I: on flow structures and turbulence," *J. Fluids Eng. - Trans ASME*, Vol. 122, pp.97-107.

[12] Sinha, M., Katz, J., and Meneveau, Ch., 2000, "Quantitative visualization of the flow in a centrifugal pump with diffuser vanes. II: addressing passage-averaged and large-eddy simulation modeling issues in turbomachinery flows," *J. Fluids Eng. - Trans ASME*, Vol. 122, pp. 108-116.

[13] Wuibaut, G., Bois, G., Dupont, P., Caignaert, G., and Stanislas, M., 2002, “PIV measurements in the impeller and the vaneless diffuser of a radial flow pump in design and off-design operating conditions,” *J. Fluids Eng. - Trans ASME*, Vol. 124 (3), pp.791-797.

[14] Pedersen, N., Larsen, P. S., and Jacobsen, C. B., 2003, “Flow in a centrifugal pump impeller at design and off-design conditions—Part I: Particle Image Velocimetry (PIV) and Laser Doppler Velocimetry (LDV) measurements,” *J. Fluids Eng. - Trans ASME*, Vol 125, pp. 61-72.

[15] Wu, Y. L., Liu, S. H., Yuan, H. J., et al., 2001, “PIV measurement on internal instantaneous flows of a centrifugal pump,” *Sci China Tech Sci.*, Vol. 54, pp.270-276.

[16] Feng, J., Benra, F. K., and Dohmen, H. J., 2009, “Unsteady flow visualization at part-load conditions of a radial diffuser pump: by PIV and CFD,” *J. Visualization*, Vol. 12 (1), pp. 65-72.

[17] Feng, J., Benra, F. K., Dohmen, H. J., 2010, “Application of different turbulence models in unsteady flow simulations of a radial diffuser pump,” *Forschung im Ingenieurwesen*, Vol. 74 (3), pp. 123-133.

[18] Feng, J. J., Benra, F. K., and Dohmen, H. J., 2011, “Investigation of periodically unsteady flow in a radial pump by CFD simulations and LDV measurements”, *J. Turbomach. - Trans ASME Trans ASME*, Vol. 133, pp.011004-11.

[19] Nishino, T., Roberts, G.T. and Zhang, X., 2008, “Unsteady RANS and Detached-Eddy Simulations of flow around a circular cylinder in ground effect”, *J. Fluids Struct.*, Vol. 24, pp. 18-33.

[20] Zhou, L., Shi, W.D., Li, W., et al, 2014, “Particle Image Velocimetry Measurements and performance experiments in a compact return diffuser under different rotating speed”, *Experimental Techniques*, online.

[21] Zhou, L., Shi, W.D., Cao, W.D., et al, 2015, “CFD investigation and PIV validation of flow field in a compact return diffuser under strong part-load conditions”, *Sci China Tech Sci.*, Vol. 58, pp. 405-414.

[22] Bai, L., Shi, W.D., Zhou, L., et al, 2013, “Numerical Simulation and Uncertainty Estimation in a Deep well Centrifugal Pump”, *IOP Conference Series: Materials Science and Engineering, IOP Publishing*, Vol. 52, pp. 22-27.

.