

Research Article

Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions

K. W. Cheah, T. S. Lee, S. H. Winoto, and Z. M. Zhao

Department of Mechanical Engineering, National University of Singapore, 9 Engineering Drive 1, Singapore 117576

Received 8 June 2006; Revised 15 January 2007; Accepted 10 April 2007

Recommended by Jiun-Jih Miao

The current investigation is aimed to simulate the complex internal flow in a centrifugal pump impeller with six twisted blades by using a three-dimensional Navier-Stokes code with a standard $k-\epsilon$ two-equation turbulence model. Different flow rates were specified at inlet boundary to predict the characteristics of the pump. A detailed analysis of the results at design load, Q_{design} , and off-design conditions, $Q = 0.43 Q_{\text{design}}$ and $Q = 1.45 Q_{\text{design}}$, is presented. From the numerical simulation, it shows that the impeller passage flow at design point is quite smooth and follows the curvature of the blade. However, flow separation is observed at the leading edge due to nontangential inflow condition. The flow pattern changed significantly inside the volute as well, with double vortical flow structures formed at cutwater and slowly evolved into a single vortical structure at the volute diffuser. For the pressure distribution, the pressure increases gradually along streamwise direction in the impeller passages. When the centrifugal pump is operating under off-design flow rate condition, unsteady flow developed in the impeller passage and the volute casing.

Copyright © 2007 K. W. Cheah et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

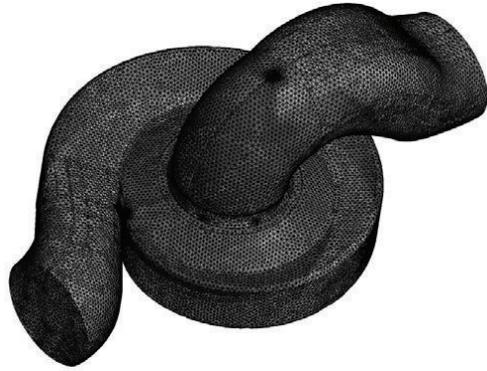
1. INTRODUCTION

The complex flow pattern inside a centrifugal pump is strong three-dimensional with recirculation flows at inlet and exit, flow separation, cavitations, and so on. The curvature of the blades and the rotational system have great influence on the flow field. Liu et al. [1], Akhras et al. [2], and Pedersen et al. [3] have made the measurement on centrifugal pumps and reported that impeller flow separation was observed on blade surface at off-design flow rate as compared to smooth flow within the impeller passage at design point. The numerical simulation made by Heilmann and Siekmann [4] and Majidi and Siekmann [5] showed the strong secondary flow in volute and circular casings of centrifugal pumps. Ziegler et al. [6], Shi and Tsukamoto [7], Shum et al. [8], and Akhras et al. [9] studied impeller diffuser interaction on the pump performance and showed that a strong pressure fluctuation is due to the unsteadiness of the flow shedding from impeller exit. Hong and Kang [10] and Hagelstein et al. [11] investigated the flow field at the impeller exit and volute separately to study the pressure distribution due to impeller-volute interaction.

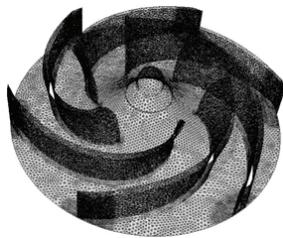
Traditional method to design the centrifugal pump is mainly based on the steady-state theory, empirical correlation, combination of model testing, and engineering exper-

ience [12]. However, to further improve the pump performance for design and off-design operating conditions, it will become extremely difficult. Complex flow field such as the boundary layer separation, vortex dynamics, interactions between the impeller and diffuser are difficult to control due to the rotating and stationary components. Zhang et al. [13, 14] found that jet-wake structure occurs near the outlet of the impeller and it is independent of flow rate and locations. Byskov et al. [15] investigated a six-bladed impeller with shroud by using the large eddy simulation (LES) at design and off-design conditions. At design load, the flow field inside the impeller is smooth and with no significant separation. At quarter design load, a steady nonrotating stall phenomenon is observed in the entrance and a relative eddy is developed in the remaining of the passage. Gu et al. [16] also investigated the volute/diffuser interaction of a single stage centrifugal compressor at design point and off-design. At higher flow rate, a twin vortex structure is formed downstream of the passage. The recirculation and the twin vortex structure are attributed increase of the total pressure losses at off-design conditions.

Hence with the advancing of computer power, significant improvement of numerical algorithms and more reliable CFD codes, it can be seen that there is an increasing trend of applying numerical methods to study the complex flow



(a)



(b)

FIGURE 1: (a) Unstructured mesh for the centrifugal pump. (b) Impeller mesh.

in a centrifugal and to improve the efficiency. For examples, Asuaje et al. [17] and Goto et al. [18] have analyzed flow in centrifugal pump impeller by using an inversed design method.

1.1. Centrifugal pump and test conditions

The centrifugal pump considered in this study consists of an impeller shrouded with six backswept blades, a curved intake section, and a spiral volute casing. The leading edge of the impeller blade is twisted with blade inlet angle α_{shroud} of 40° at shroud and α_{hub} of 35° at hub. The impeller blade trailing edge is straight with blade outlet angle β_2 of 23° . The impeller outlet diameter d_2 is 356 mm, with an outlet width b_2 of 46.8 mm. The flow from impeller is discharged into a spiral volute casing with mean circle diameter d_3 of 374 mm. The impeller is designed to operate at 1450 rpm with the specific speed as 131. A detailed description of the experimental set up and measurement procedures can be found from the work done by Zhao [19]. The pressure transducers are used at inlet and outlet to measure the pressure head across the pump. The volume flow rate is controlled by a valve and is measured by a magnetic flow meter at the downstream.

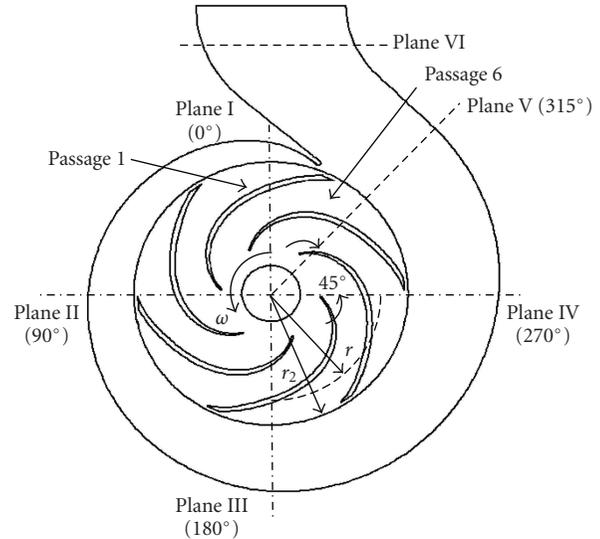


FIGURE 2: Cross-sectional plane location at volute casing.

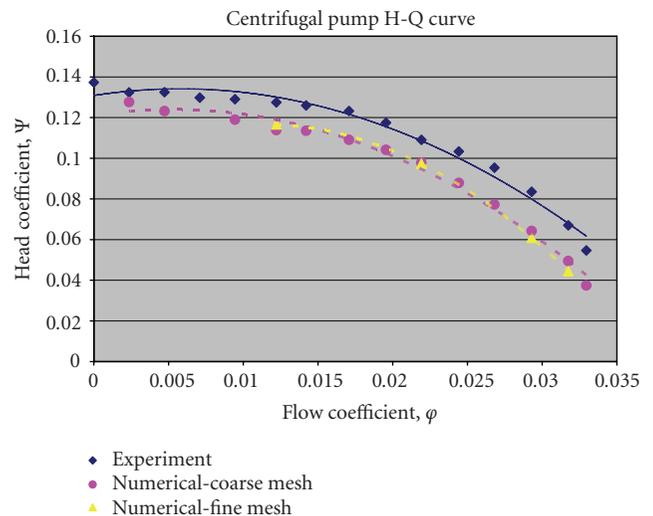


FIGURE 3: Comparison of numerical and experiment characteristics curves.

1.2. Numerical technique

At present study, the commercially available CFD code, CFX is used. The $k-\epsilon$ standard turbulence model is used and the walls are modeled using a log-law wall function. Validation of this code can be found from the work done by Majidi and Siekmann [5], Gu et al. [16], and Zangeneh et al. [20] that have used it in turbomachinery applications. The current numerical computation is carried out with a multiple frame of reference approach because the impeller flow field is within a rotating frame whereby the intake section and volute casing are in stationary frame. The meshes of three computational domains, the intake section, impeller, and volute casing, are generated separately. The unstructured mesh with tetrahedral element is being used in all three computational

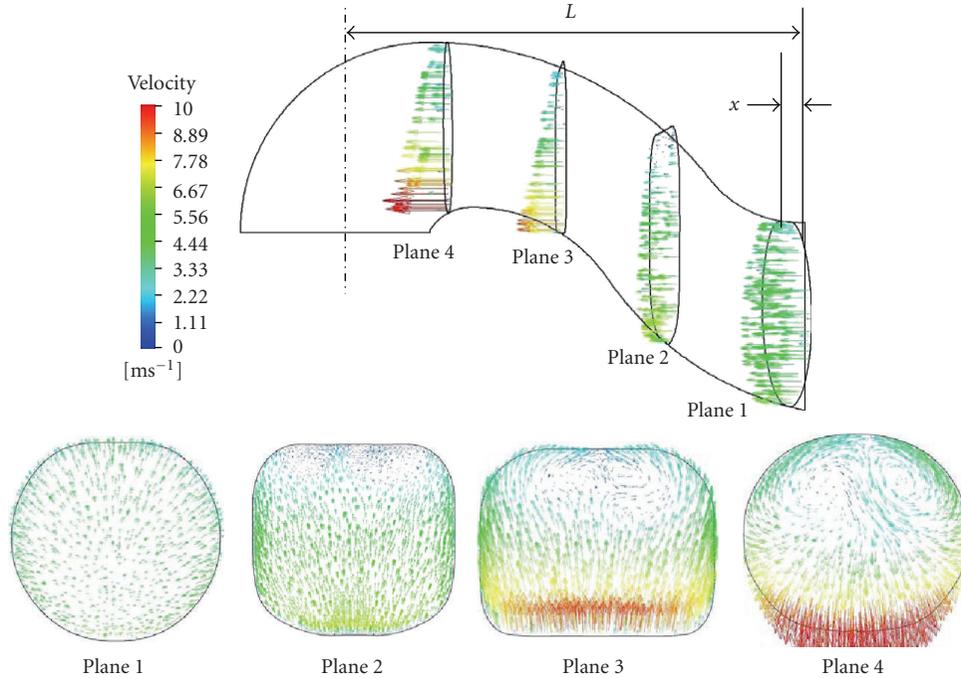


FIGURE 4: Flow separation in intake section at various location.

domains. A localized refinement of mesh is employed at regions close to volute tongue area, impeller blade leading and trailing edge in order to accurately capture the flow field structure. This is because the flow field properties variation such as pressure and velocity at these regions are expected to be substantial. The intake section and volute domains consist of 600 913 and 818 423 tetrahedral elements, respectively, while impeller domain consists of 100 4139 elements. The number of elements used in the numerical simulation is fixed after the grid independence study. Grid independence study results will be discussed in later section. Figure 1 shows the grid assembly of intake, impeller, and volute sections. In this numerical simulation, the meshes of the impeller and the volute casing are connected by means of a frozen-rotor interface. The two frames of reference, rotating and stationary, are connected in such a way that for steady-state calculations the relative position of the impeller and volute casing does not change through the calculations.

Figure 2 shows the six cross-sectional planes that cut in according to the various locations in volute casing for later discussion. With 0° , plane is closest to volute tongue and advancing in angular direction up to 315° . The impeller passages are labeled from 1 to 6 in anticlockwise direction with Passage 1 closest to the volute tongue.

Present numerical computation is based on steady-state condition with the following boundary conditions imposed: at the inlet of the computational domain the mass flow rate, the turbulence intensity, and a reference pressure are specified. The mass flow rates are specified ranging from $1.45 Q_{\text{design}}$, Q_{design} , and up to $0.43 Q_{\text{design}}$. The absolute velocity at the inlet is defined in such a way that is perpendicular

to the inlet mesh surface. Turbulent intensity was specified to be 5%. At the outlet (end of volute diffuser), mass flow is being specified as well. All the other variables are free to float. The volute casing and intake section walls are in stationary frame and modeled using a no-slip boundary condition.

2. RESULTS AND DISCUSSION

Prior to any discussion of the numerical results, a comparison of the numerical and experimental performance curves for the pump is shown in Figure 3. For the numerical simulation, grid independence test has been performed. At design point, the head coefficient for fine mesh is 0.097 as compared to 0.096 for coarse mesh. Hence, the numerical results show no significant difference between the coarse and fine mesh. The numerically calculated head follow the trend very well as compared to the experimental ones. However, it could not match the experimental head one-to-one as reported by the work done by González et al. [21], but it has similar accuracy as predicted by Byskov et al. [15] and Meakhail and Park [22].

2.1. Velocity field

At the intake section, Figure 4, the flow separation can be clearly seen on the top wall region. The distance from the inlet to the center of the impeller, L , is 500 mm. The dimensionless distances of Plane 1 to 4, x/L , to the center of the impeller are 0.04, 0.29, 0.50, and 0.74, respectively. The flow separation-developed upstream has great influence to the flow entering impeller eye. Starting from Plane 1, the flow is very smooth and uniform. As the flow approaching to

the bended section at Plane 2, the flow at top wall region becomes unstable and separating. Further downstream at Plane 3 and 4, the vortical flow structures predominantly developed near the top wall region. The flow in this kind curved conduit is known to be prompt to flow separation and is well documented in literature. As noticed in Plane 4, the vortical flow structure will propagate further downstream to impeller eye.

As the flow entering the impeller eye, it is diverted into the blade-to-blade passage. Due to the unsteady effect developed at upstream, the flow entering the passage is no longer tangential to the leading edge of impeller blade. Those so-called shockless velocity entries to impeller passage will no longer be guaranteed. Separation of flow can be observed at all passages leading edge. Figure 5 shows the leading edge velocity vector field with separation and inflow incident angle that is nontangent to the blade leading edge. At three different flow rates, the leading edge flow separation patterns are slightly different. This could be contributed by the boundary layer thickness that grows from the leading edge at suction side with different rates. The flow separation could stretch up to 10% of the blade cord length downstream. This inlet flow condition at impeller passage could be the possible source of cavitations. Coutier-Delgosha et al. [23] predicted that the cavitations at leading edge with various NPSH by numerical methods compared well with experimental measurement at nominal flow rate. This leading edge separation could lead to energy loss in the pump and could further influence the flow field in impeller passage in streamwise direction.

The impeller passage flow at design point is very smooth and well-guided except at the leading edge as discussed above. The flow follows the blade curvature profile from impeller passage entrance till the exit without any separation on blade pressure side. This is matching the potential flow theory for flow pass turbomachinery blade.

When the centrifugal pump is running with $Q/Q_{\text{design}} = 0.43$, a strong recirculation flow developed inside the impeller passage. The center region of the blade-to-blade passage near the suction side has a low velocity region and is considered as stall flow. This phenomenon could be considered as “jet-wake” structure development phase as reported by many researchers [2, 14, 15]. The jet-wake structure is found to be caused from leading edge flow separation on suction side. At $Q/Q_{\text{design}} = 1.45$, the velocity vector inside impeller passage is also very smooth and no separation of flow is observed. The recirculation flow at leading edge remains there.

2.2. Unsteady flow developed in volute casing

When the flow within the impeller passage being discharge into the volute casing, Figure 6, a strong recirculation flow is developed. Near volute tongue (Plane 1), twin vortices are formed. As the flow make angular advancement according to the cross-sectional plane with $\theta = 90^\circ, 180^\circ, 270^\circ, 315^\circ$, and finally at the exit plane (Plane VI, parallel to outlet), the evolution of the bottom vortex core is very prominent and dominant. The intensity of the lower vortex is increasing as the

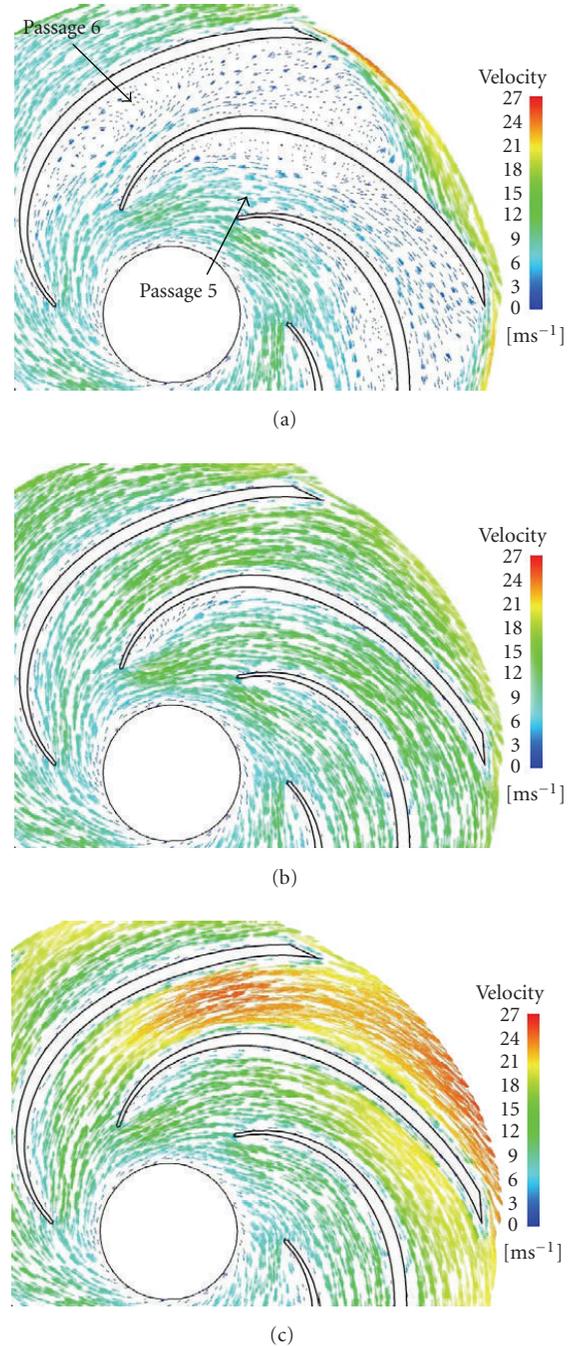


FIGURE 5: Relative velocity inside impeller passage 5 and 6. (a) Off-design point, $Q = 0.43 Q_{\text{design}}$. (b) Design point, Q_{design} . (c) Off-design point, $Q = 1.45 Q_{\text{design}}$.

flow approaching exit plane and this could be corresponding to the energy or head loss in a pump. The single and double vortical flow structures inside the volute casing has been reported by many researchers. Investigation done by Nursen and Ayder [24] on an external-type volute with a rectangular cross-sectional shape that having a constant axial width and also investigation by Majidi and Siekmann [5] with single volute casing that is designed according to the theory of a

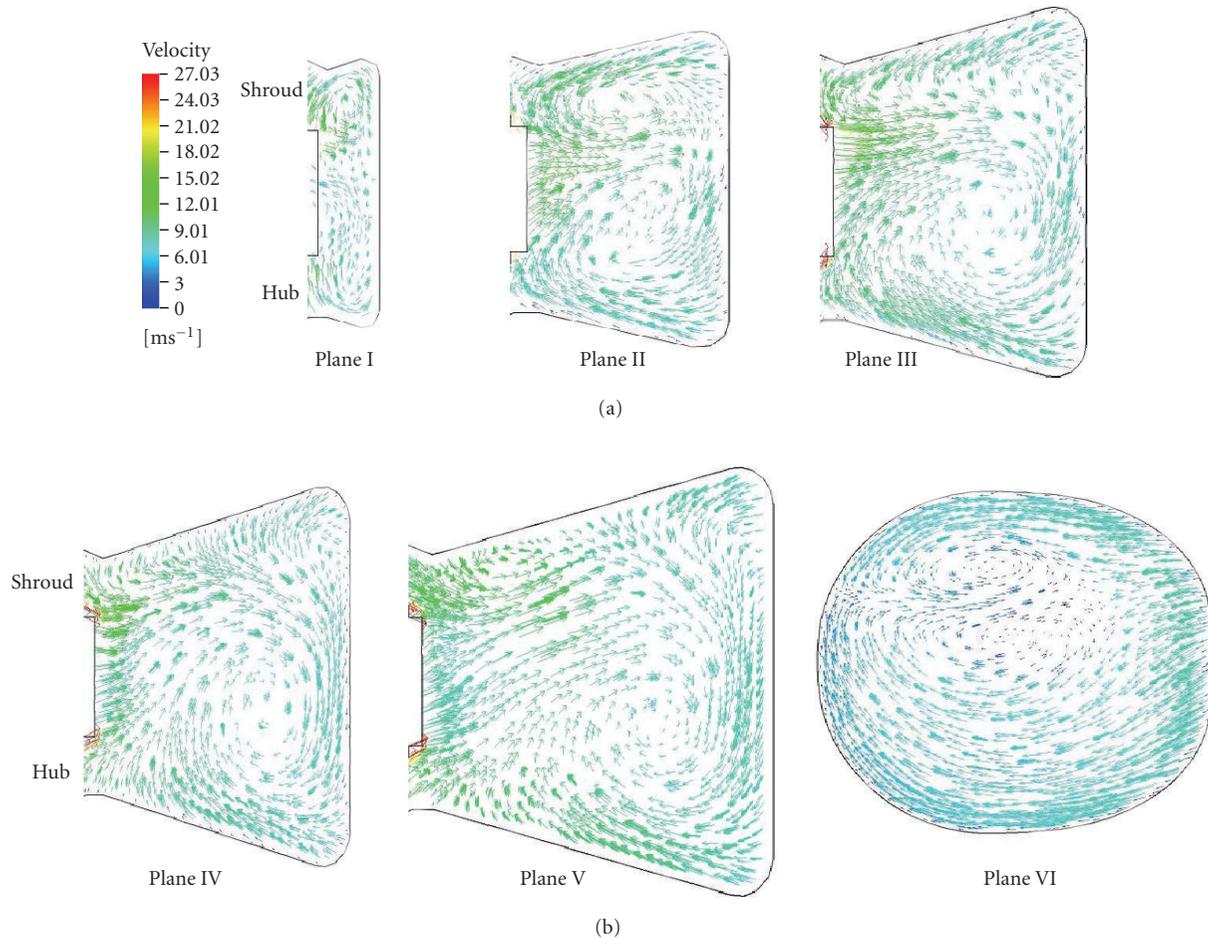


FIGURE 6: Unsteady flow developed inside the volute casing at various locations.

constant average velocity for all sections of the volute showed the development of vortical flow structures within the volute casing. Based on the vortices pattern formed inside the volute, the secondary flow inside the volute is sensitive to the volute geometry and the jet-wake structure from the impeller passage. The relative gaps between the shroud and volute casing, hub and volute casing, will have influence on the secondary flow formation as well in streamwise direction. This is because there will be always a back flow (leakage) from the gaps between hub, shroud, and volute casing. The work done by Khelladi [25] indicated that leakage flow between axial clearance of impeller and volute casing caused the pressure fluctuation and flow recirculation at the impeller exit.

However, at different flow rate conditions, the unsteady flow at exit plane shows a significant different pattern as well. Figure 7 shows the flow pattern at the volute diffuser with different flow rate. At Q_{design} , the flow collected in the volute casing flowing out smoothly at volute diffuser section. As compared to lower flow rate, with lack of momentum, a strong recirculation flow at volute diffuser section is observed. At the region close to volute tongue, a significant backflow is reentering passage 6. At $Q/Q_{\text{design}} = 1.45$, the flow on the outer wall of volute casing is accelerating out as

compared with the stalled flow at inner wall (close to volute tongue region). The high momentum main flow tends to follow the curvature of the volute outer wall with flow at region behind the volute tongue is blocked and stalled. This can be considered as the flow with high velocity, from impeller passage 6 discharged as jet into volute with flow behind the volute tongue is almost stagnant.

2.3. Pressure distribution and blade loading

When the pump is operating at design point, Figure 8 shows the pressure distribution within the impeller and volute casing. The pressure increases gradually along streamwise direction within impeller blade-to-blade passage and has higher pressure on pressure surface than suction surface for each plane. However, the pressure developed inside the impeller and volute is not so uniform. The isobar lines are not all perpendicular to the pressure side of the blade inside the impeller passage, this indicated that there could be a flow separation because of the pressure gradient effect.

The pressure loading on the impeller blade can be seen in Figure 9. The pressure load on the impeller blade is plot against the normalized radial direction with r/r_2 . The

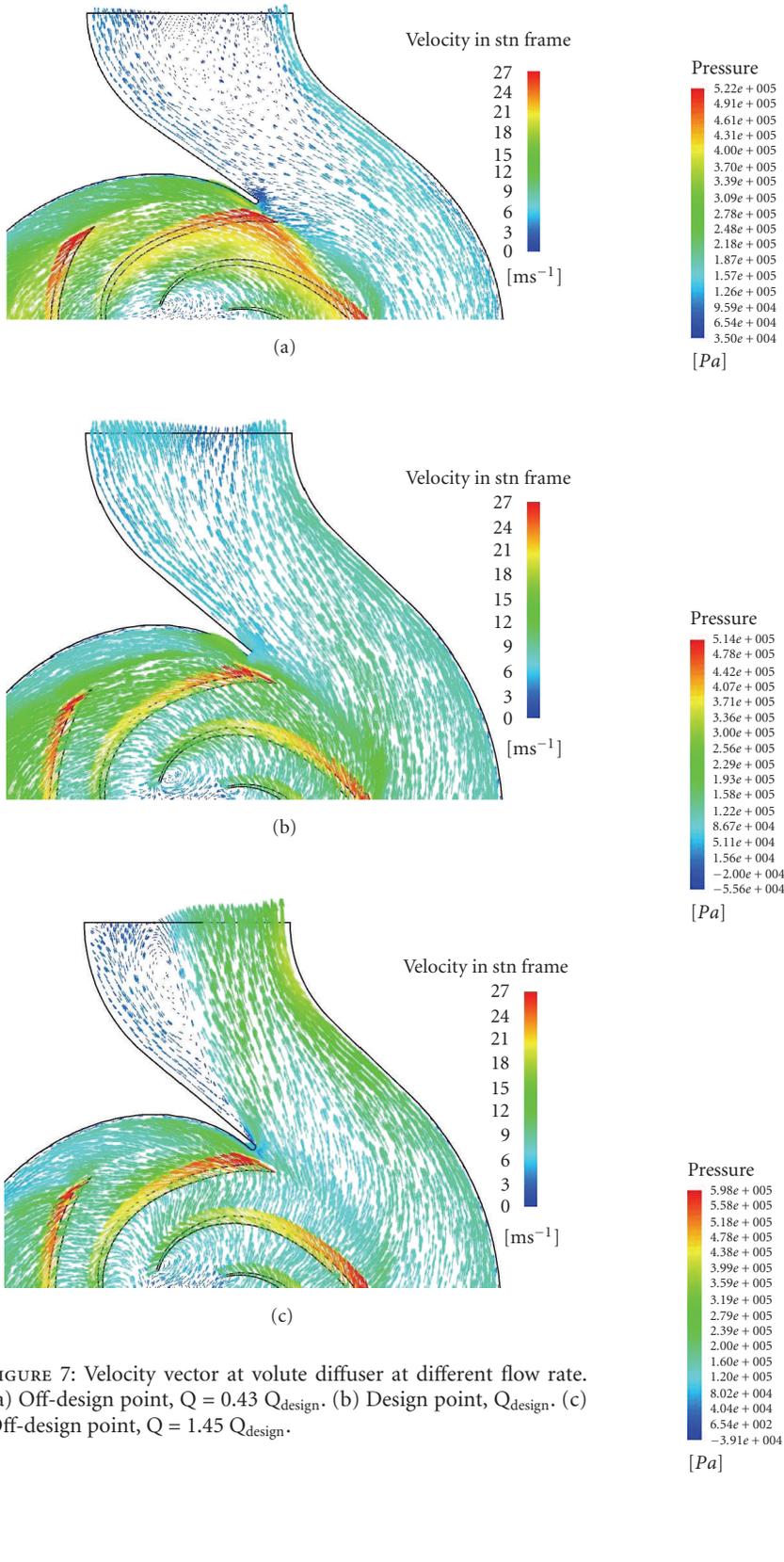


FIGURE 7: Velocity vector at volute diffuser at different flow rate. (a) Off-design point, $Q = 0.43 Q_{design}$. (b) Design point, Q_{design} . (c) Off-design point, $Q = 1.45 Q_{design}$.

pressure load does not increase linearly along the cord length direction on either side of impeller blade. The pressure difference on the pressure and suction sides on the blade at mid-chord length ($r/r_2 = 0.6$) and near impeller exit suggested

FIGURE 8: Pressure distribution inside impeller and volute. (a) Off-design point, $Q = 0.43 Q_{design}$. (b) Design point, Q_{design} . (c) Off-design point, $Q = 1.45 Q_{design}$.

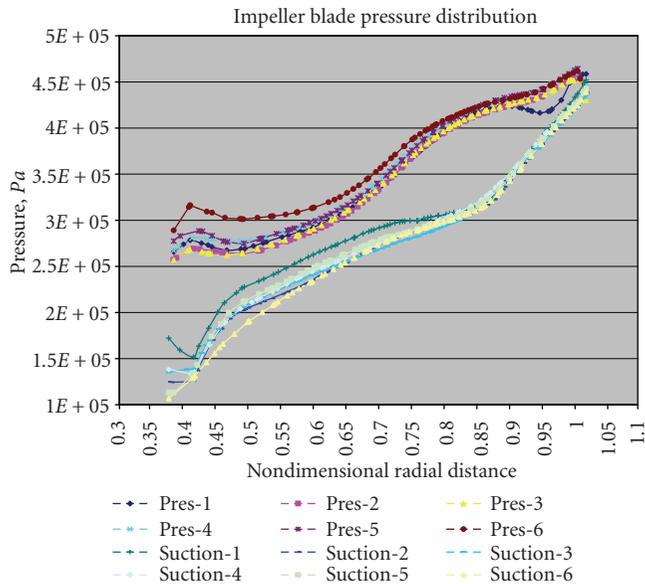


FIGURE 9: Pressure loading on impeller at Q_{design} .

that the flow inside the impeller passage experience shearing effects due to the pressure difference on blade-to-blade passage wall.

3. CONCLUSION

The complex pump internal flow field is investigated by using numerical methods and compared well with experimental data over the wide flow range. The comparison of the predicted head and efficiency between numerical results and experiment data shows a good agreement. The complex internal flow of the centrifugal pump simulation has permitted to study the internal flow pattern and pressure distribution of the pump operating at design point and off-design point.

At design point, the internal flow or velocity vector is very smooth along the curvature along the blades. However, flow separation developed at the leading edge due to nontangential inflow conditions. The single and double vortical flow structures are observed in the volute casing.

When operating at off-design load, the flow pattern has changed significantly from the well-behaved flow pattern at design load condition. A strong flow recirculation at the center of the passage of the impeller can be observed. The stall region developed due to the recirculation is blocking the flow passing through the passage. In this case, the rotational effect is also an important factor to be considered.

As for the pressure distribution, the pressure increases gradually along the streamwise direction. The pressure lines are seen to be inclined in the circumferential direction. It is also found that the isobars are no longer perpendicular to the impeller suction surface at low flow rate. The static pressure drops severely as impeller rotational speed decreases.

REFERENCES

- [1] C. H. Liu, C. Vafidis, and J. H. Whitelaw, "Flow characteristics of a centrifugal pump," *ASME Journal of Fluids Engineering*, vol. 116, no. 2, pp. 303–309, 1994.
- [2] A. Akhras, M. El Hajem, R. Morel, and J.-Y. Champagne, "Internal flow investigation of a centrifugal pump at the design point," *Journal of Visualization*, vol. 4, no. 1, pp. 91–98, 2001.
- [3] N. Pedersen, P. S. Larsen, and C. B. Jacobsen, "Flow in a centrifugal pump impeller at design and off-design conditions—part I: particle image velocimetry (PIV) and laser Doppler velocimetry (LDV) measurements," *ASME Journal of Fluids Engineering*, vol. 125, no. 1, pp. 61–72, 2003.
- [4] C. M. Heilmann and H. E. Siekmann, "Particle image velocimetry as CFD validation tool for flow field investigation in centrifugal pumps," in *Proceedings of the 9th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC '02)*, Honolulu, Hawaii, USA, February 2002.
- [5] K. Majidi and H. E. Siekmann, "Numerical calculation of secondary flow in pump volute and circular casings using 3D viscous flow techniques," *International Journal of Rotating Machinery*, vol. 6, no. 4, pp. 245–252, 2000.
- [6] K. U. Ziegler, H. E. Gallus, and R. Niehuis, "A study on impeller-diffuser interaction—part I: influence on the performance," *Journal of Turbomachinery*, vol. 125, no. 1, pp. 173–182, 2003.
- [7] F. Shi and H. Tsukamoto, "Numerical study of pressure fluctuations caused by impeller-diffuser interaction in a diffuser pump stage," *ASME Journal of Fluids Engineering*, vol. 123, no. 3, pp. 466–474, 2001.
- [8] Y. K. P. Shum, C. S. Tan, and N. A. Cumpsty, "Impeller-diffuser interaction in a centrifugal compressor," *Journal of Turbomachinery*, vol. 122, no. 4, pp. 777–786, 2000.
- [9] A. Akhras, M. El Hajem, J.-Y. Champagne, and R. Morel, "The flow rate influence on the interaction of a radial pump impeller and the diffuser," *International Journal of Rotating Machinery*, vol. 10, no. 4, pp. 309–317, 2004.
- [10] S.-S. Hong and S.-H. Kang, "Flow at the centrifugal pump impeller exit with circumferential distortion of the outlet static pressure," *ASME Journal of Fluids Engineering*, vol. 126, no. 1, pp. 81–86, 2004.
- [11] D. Hagelstein, K. Hillewaert, R. A. Van den Braembussche, A. Engeda, R. Keiper, and M. Rautenberg, "Experimental and numerical investigation of the flow in a centrifugal compressor volute," *Journal of Turbomachinery*, vol. 122, no. 1, pp. 22–31, 2000.
- [12] A. J. Stepanoff, *Centrifugal and Axial Flow Pumps: Theory, Design and Application*, Krieger, Melbourne, Fla, USA, 2nd edition, 1992.
- [13] M.-J. Zhang, M. J. Pomfret, and C. M. Wong, "Three-dimensional viscous flow simulation in a backswept centrifugal impeller at the design point," *Computers and Fluids*, vol. 25, no. 5, pp. 497–507, 1996.
- [14] M.-J. Zhang, M. J. Pomfret, and C. M. Wong, "Performance prediction of a backswept centrifugal impeller at off-design point conditions," *International Journal for Numerical Methods in Fluids*, vol. 23, no. 9, pp. 883–895, 1996.
- [15] R. K. Byskov, C. B. Jacobsen, and N. Pedersen, "Flow in a centrifugal pump impeller at design and off-design conditions—part II: large eddy simulations," *ASME Journal of Fluids Engineering*, vol. 125, no. 1, pp. 73–83, 2003.

- [16] F. Gu, A. Engeda, M. Cave, and J.-L. Di Liberti, "A numerical investigation on the volute/diffuser interaction due to the axial distortion at the impeller exit," *ASME Journal of Fluids Engineering*, vol. 123, no. 3, pp. 475–483, 2001.
- [17] M. Asuaje, F. Bakir, S. Kouidri, and R. Rey, "Inverse design method for centrifugal impellers and comparison with numerical simulation tools," *International Journal of Computational Fluid Dynamics*, vol. 18, no. 2, pp. 101–110, 2004.
- [18] A. Goto, M. Nohmi, T. Sakurai, and Y. Sogawa, "Hydrodynamic design system for pumps based on 3-D CAD, CFD, and inverse design method," *ASME Journal of Fluids Engineering*, vol. 124, no. 2, pp. 329–335, 2002.
- [19] Z. M. Zhao, "Design of centrifugal pump using computational fluid dynamics," M.Eng thesis, National University of Singapore, Singapore, 2002.
- [20] M. Zangeneh, M. Schleer, F. Pløger, et al., "Investigation of an inversely designed centrifugal compressor stage—part I: design and numerical verification," *Journal of Turbomachinery*, vol. 126, no. 1, pp. 73–81, 2004.
- [21] J. González, J. Fernández, E. Blanco, and C. Santolaria, "Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump," *ASME Journal of Fluids Engineering*, vol. 124, no. 2, pp. 348–355, 2002.
- [22] T. Meakhail and S. O. Park, "A study of impeller-diffuser-volute interaction in centrifugal fan," *Journal of Turbomachinery*, vol. 127, no. 1, pp. 84–90, 2005.
- [23] O. Coutier-Delgosha, R. Fortes-Patella, J. L. Reboud, M. Hofmann, and B. Stoffel, "Experimental and numerical studies in a centrifugal pump with two-dimensional curved blades in cavitating condition," *ASME Journal of Fluids Engineering*, vol. 125, no. 6, pp. 970–978, 2003.
- [24] E. C. Nursen and E. Ayder, "Numerical calculation of the three-dimensional swirling flow inside the centrifugal pump volutes," *International Journal of Rotating Machinery*, vol. 9, no. 4, pp. 247–253, 2003.
- [25] S. Khelladi, S. Kouidri, F. Bakir, and R. Rey, "Flow study in the impeller-diffuser interface of a vaned centrifugal fan," *ASME Journal of Fluids Engineering*, vol. 127, no. 3, pp. 495–502, 2005.



Hindawi

Submit your manuscripts at
<http://www.hindawi.com>

