Unsteady Fluid Flow Study in a Centrifugal Pump

by CFD Method

K.W Cheah*, T.S. Lee, and S.H Winoto
Department of Mechanical Engineering,
National University of Singapore,
Block EA, 07-08, 9 Engineering Drive 1,
Singapore 117576
Tel: +65-6516-2212 / FAX: 65-6516-4498
*E-mail: g0500401@nus.edu.sg

Abstract

Unsteady flow field within a centrifugal pump is very complex and three dimensional. Present investigation within a centrifugal pump with six twisted blade impeller is to understand the impeller and volute tongue interactions. The numerical analysis is done by solving the three-dimensional RANS codes with standard $k-\varepsilon$ two-equations turbulence model. Wall regions of the computational domain are modeled with a scalable log-law wall function. Current numerical modeling is done with multiple frames of reference and the dissimilar tetrahedral mesh interface of the impeller/volute casing is connected with sliding mesh technique. The analysis shows that there is a recirculation zone near to suction-front shroud side just after the leading edge even at design point. However, the flow within the impeller passage is very smooth and following the curvature of the blade in stream-wise direction. Jet/wake flow pattern that commonly occurs in centrifugal impeller also observed within the impeller passage. When the flow is discharged into volute casing circumferentially from the impeller outlet, the high velocity flow is severely distorted and formed a spiral flow pattern within the volute casing. Near volute tongue region, the impeller/volute tongue strong interaction is observed based on the periodically fluctuating pressure at outlet. The results of existing analysis proved that the pressure fluctuation periodically is due to the position of impeller blade relative to tongue and the flow field within the volute casing is always unsteady and turbulent.

Keywords: centrifugal pump, impeller, unsteady flow, spiral flow

1. INTRODUCTION

Centrifugal pump is one of the main drivers in industrial revolution. However, modern centrifugal pump design still depends heavily on testing and experiment. This is because the flow pattern inside the centrifugal pump is highly turbulent, unsteady and three-dimensional. The unsteady flow phenomenon arises from strong interactions between the complex geometry of impeller and volute causes pump design even more challenging at off-design conditions. Hence, better understanding of the complex flow field inside the centrifugal pump can help pump designers achieve better design and higher efficiency pumps.

The jet/wake flow structure is a common flow phenomenon in centrifugal pump. Bwalya and Johnson [1] and Murakami et al [2] reported that leading edge flow separation on the pressure side/shroud corner of a centrifugal pump could propagate downstream to form a wake region on the suction side/shroud corner. The wake flow on the shroud suction side is attributed to the rapid change of flow direction at the passage inlet. However, the flow field within the impeller is flow rate dependent as well. Measurement made by Liu et al [3] and Pedersen et al [4] showed that the flow within the impeller passage is very smooth and follows the curvature of the blades at design point. At off-design point condition, with lower flow rate, deceleration of radial flow on shroud surface of the blades and increase of secondary flows and vorticity in the passage were observed.
Investigation by Dong et al [5] and Chu et al [6] using PIV method at design and off design conditions show the strong rotating impeller and volute casing interactions. They found the present of the jet/wake structures and pulsating flow near impeller exit. A vortex train is generated as a result of the non-uniform out-fluxes from the impeller. The strong impeller volute interaction causes unsteady flow developed inside the volute as well. As the flow discharge from the impeller exit into volute casing, it is turbulent and three-dimensional. Depending on the volute geometry, different swirling flow occurs. Elhom et al [7] LDV measurement inside a logarithmic volute showed that impeller exit radial velocity transformed into a swirling flow with a forced vortex type velocity distribution at center and a nearly constant swirl velocity away from it. However, the vortex flow center and through flow velocity center are not coincide to each other and is flow rate dependent.

The use of CFD to analyze the complex flow field inside the pumps has been increasing due to the good agreement between the numerical and experimental results. Miner [8] has demonstrated good agreement between CFD analysis of shape and magnitude of the velocity and pressure profiles with measured results for the axial and mixed flow pumps. González et al [9] successfully studied the unsteady flow structure in a pump and showed a good agreement between experimental and numerical results. Both experiments and numerical prediction show the presence of a spatial fluctuation pattern at the blade passing frequency as function of the flow rate. Investigation by Byskov and Jacobsen [10] using the large eddy simulation (LES) at design and off-design conditions showed that their numerical results are in good agreement with experimental results.

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 of 40°. The impeller blade trailing edge is straight with blade outlet angle β2 of 23°. The impeller outlet diameter d2 is 356 mm and outlet width, b2 of 46.8 mm. The flow from impeller is discharged into a spiral volute casing with mean circle diameter r3 of 374 mm. The pump is designed to operate at 1450 rpm with the design flow rate, Q_design of 600 m³/hr and pump head, H of 30 m. The Reynolds number for current study ρND2/μ = 2.15E7.

2. NUMERICAL MODELING

The commercially available CFD code [11], CFX 11.0 with standard k-ε turbulence model is used and the walls are modeled using a scalable log-law wall function. Current numerical computation is carried out in two parts, quasi-steady and unsteady. Quasi-steady numerical computation is carried out with a multiple frames of reference (MFR) approach in which the impeller flow field is with reference to a rotating frame whereby the volute casing and intake section refer to a stationary frame. Figure 1 shows the pump assembly mesh and the impeller mesh. The tetrahedral elements of intake section, impeller and volute are generated separately, so the dissimilar meshes are connected by means of a Frozen-Rotor interface. Although Frozen Rotor interface is mainly used for the axis-symmetric problem, but the fast convergence of this model can save huge computation time to obtain the overall pump performance curve.

Figure 1 Pump assembly and impeller mesh

The numerical computation is carried out with the following boundary conditions imposed: At the inlet of the intake section the total pressure, the turbulence intensity and a reference pressure are specified. The absolute velocity vector at the inlet is defined in such a way that is perpendicular to the inlet mesh surface. Turbulent intensity was specified to be 5%. Mass flow rate is prescribed at volute outlet. Wide range of mass flow rate is specified in order to numerically predict the pump characteristics. All the other variables are free to float. Solid walls of the impeller blades, hub and shroud in rotating frame and the walls of the volute casing and intake are in stationary frames and modeled using a no-slip boundary condition. The numerical computation is considered converged when the maximum residual 10E-4 is reached.
For the unsteady calculation, the dissimilar mesh at the intake, impeller and volute interface are connected by means of Transient Rotor/Stator interface. Due to computation time and resource constraints, transient (unsteady) analysis is only performed at selected flow rate, \( Q/Q_{\text{design}} = 0.70 \), \( Q/Q_{\text{design}} = 1 \) and \( Q/Q_{\text{design}} = 1.2 \). The quasi-steady result is used to initialize the unsteady calculation. The boundary conditions are the same as the quasi steady state calculation. The time step of the unsteady calculation has been set to 6.8966E-4 seconds. This time step size is corresponding to 6 degree per time step for the rotational speed of the impeller of 1450 rpm. One complete impeller revolution is performed after 60 time steps. The total number of time step 660, which is equal to 11 revolutions of the impeller and the total time is 0.45517 s.

Figure 2 shows the plan-view of the pump and the mid-plane is located at \( z/b = 0.5 \). Eight cross-sectional planes are cut in according to the various angular locations in volute casing for later discussion. With Plane I at 0° is closest to volute tongue and the following Plane II to Plane VIII with an increment of 45° in anti-clockwise angular direction up to 315°. Plane IX is 350 mm away from Plane III-VII. The impeller passages are labeled from 1 to 6 in anti-clockwise direction with Passage 1 closest to the volute tongue. Similarly, the impeller blades are labeled as Blade 1 to 6 in anti-clockwise direction with Blade 1 is between Passage 1 and 6, Blade 2 is between Passage 1 and 2, and so on.

3. RESULTS AND DISCUSSION

The numerically predicted head coefficient over wide flow range using quasi-steady approach is compared well and in good agreement with the experimental result as shown in Figure 3.

![Figure 3 Numerical and Experimental Pump Performance Curves.](image)

At the design volume flow rate \( Q_{\text{design}} \) of 600 m³/hr, the computed steady head coefficient is only 3.8% less than the experimental result. The transient (unsteady) numerical computation head coefficient is 0.114, 0.097 and 0.074 respectively at three different flow rates of \( Q/Q_{\text{design}} = 0.70 \), \( Q/Q_{\text{design}} = 1 \) and \( Q/Q_{\text{design}} = 1.2 \) and showed a similar trend as well with quasi-steady and experimental pump performances curve. This suggested that the quasi-steady computation approach can be used to obtain the overall pump performance in less computation time yet give a reasonable accuracy.

Flow Field at Intake Section

The intake section is designed in such a way that it has a non-circular but constant cross-sectional area. The inlet section connecting to upstream pipe is circular and progressively change to a corner-rounded rectangle section at mid-span and becomes circular again just before the impeller eye. A straight partition vane is located at the middle intake section before the impeller eye and is not shown here. The distance from the center of the impeller to the intake section just before two times of intake diameter extension, \( L \), is 500 mm. The dimensionless distances of Plane I to 3, \( x/L \), to the center of the impeller are 0.25, 0.45 and 0.6 respectively.
At the intake section, as shown in Figure 4, the flow separation can be clearly seen on the top wall region due to the influence of the geometry even with the uniform inflow. The flow is very smooth and uniform starting from inlet up to Plane 3, not shown here, where the curvature of the top wall starts to influence the flow field. As the flow approaching to the bended section at Plane 2, the flow at top wall region detached and becomes unstable. This is clearly shown in Plane 2 where the low velocity twin cores developed due to the flow separation. Further downstream at Plane 1, 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. The flow separation developed upstream has great influence to the flow entering impeller eye.

Unsteady flow in volute casing

When the flow discharged from impeller passage into the volute casing, Figure 6 a strong recirculation flow developed.

Near volute tongue (Plane I), an axisymmetric counter-rotating dual core vortex formed. The high momentum flow discharge from the impeller outlet formed the vortex flow at plane I to III because of the confined volute flow passage with rounded corners that is inducing the vortex flow at the corners. As the flow advancing in angular direction according to the cross-sectional plane with = 45°, 90°, 135°, 180°, 225°, 270°, 315° and finally at the exit plane (Plane IX, parallel to outlet), the axisymmetric dual core vortex flow evolved into an asymmetric dual core vortex flow with stronger bottom vortex core. The top vortex is being
“pushed” to the right top corner. This suggests that the flow is in helical form. The formation of the asymmetrical counter rotating vortex flow is due to the distorted velocity profile at impeller outlet. The jet/wake flow structures are common flow pattern observed in centrifugal pump passage. As the wake flow has a low velocity region near shroud/suction corner and the jet flow is a high speed region near hub/pressure side. Hence the flow exiting from impeller is experiencing a shearing effect due to the distorted velocity profile and a counter rotating vortex formed.

The lower vortex core size is increasing as the flow approaching exit plane and this could be corresponding to the energy or head loss in a pump. The single and double vortical flow structure inside the volute casing has been reported by many researchers. Investigation done by Nursen and Ayder [12] on an external type volute with a rectangular cross-sectional shape that having a constant axial width and by Majidi [13] with single volute casing that is designed according to the theory of a 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 stream wise direction. This is because there will be always a back flow (leakage) from the gaps between hub, shroud and volute casing.

**Unsteady Pressure**

The strong unsteady pressure pulsation due to impeller/volute tongue interactions is shown in Figure 7. The periodic fluctuating head coefficient is plotted against the relative angular position of an impeller blade to the volute tongue. Angular position $0^\circ$ is where the impeller trailing edge is aligned with the volute tongue. Since the impeller has six blades, the pitch of the trailing edge is $60^\circ$. Hence, at relative angular position of $30^\circ$, the volute tongue is positioned at between two trailing edges. The root mean square of unsteady head coefficient is higher as compared to quasi-steady computation one. This is because the quasi-steady method only considered the average flux at the rotating and stationary components interfaces, unlike the quasi-unsteady method update the flow fields for very times step between the rotating and stationary components interfaces.

![Figure 7 Periodic pressure head fluctuation due to impeller volute interactions.](image)

Figure 7 Periodic pressure head fluctuation due to impeller volute interactions.

![Figure 8 Local unsteady pressure distribution due to impeller relative position (a) 24°, (b) 18°, (c) 12° and (d) 0°.](image)

Figure 8 Local unsteady pressure distribution due to impeller relative position (a) 24°, (b) 18°, (c) 12° and (d) 0°.

Figure 8 illustrates how the local pressure distribution is changing within the impeller and volute casing. The pressure increases gradually along stream-wise direction within impeller blade-to-blade passage However, the isobar lines are not all perpendicular to the blade surface inside the impeller passage. This suggested that the pressure do not increase linearly and uniformly in streamwise direction. When the tongue is at $0^\circ$ relative to the volute tongue, it can be seen that the isobar contours within the blade-to-blade overlap region are parallel and perpendicular to the blade pressure and suction sides. This is in good theoretical agreement where the pressure is increase in streamwise direction within the impeller passage. However, after the blade-to-blade overlap region, or so called the “throat” area, the isobar lines no longer smooth. The curvy isobar lines are the evidence of the instantaneous fluctuation pressure at the impeller periphery as
seen in Figure 7. The volute tongue has a localized high pressure envelop due to the stagnation pressure point. As the impeller rotating in anti-clockwise direction, the curvy isobar lines wiggle around the impeller periphery line. However, these localized pressure fluctuation is affecting the global pump delivery head as well. As in Figure 7 the pump delivery head is dependent on the location of the impeller trailing edge relative position with the volute tongue.

4. CONCLUSIONS

The complex and unsteady flow field within a centrifugal pump is investigated by using computational method with a standard $k$ -- $\varepsilon$ two equation model and compared well with experimental data over wide range of volume flow rate.

Recirculation developed at intake section due to the curvature effect has influenced flow into impeller eye. The flow into impeller passage is deviated from ideal and shockless condition can caused leading edge flow separation. At design point, the internal flow within impeller passage is very smooth along the curvature of the blades except with a weak recirculation at the leading edge suction/shroud corner. When the flow discharged from impeller into volute casing, spiraling vortex flow can be observed. The axisymmetric vortex flow near volute tongue evolved into an asymmetric vortex at volute outlet. The core size of the vortex flow also changing as the vortex flow advanced in angular direction, from equal core size near volute tongue to a diagonally large bottom core size. The existing analysis proved that the pressure fluctuation periodically is due to the position of impeller blade relative to tongue and the flow field within the volute casing is always unsteady and turbulent.

With the numerical simulation, it is proved to be useful for analyzing the complex flow field in the impeller and provides an improved insight into the fluid dynamics with a satisfactory accuracy compared with actual data. This is particularly important where the off-design load flow is highly unsteady.

REFERENCES


