Postprint: Study on Internal Flow Characteristics of Centrifugal Pumps under Off-Design Conditions

Authors: Gao Bo, Wang Xingning, Li Zhong, Zhang Ning, Yang Minguan

Date: 2017-11-07T00:00:00+00:00

Abstract

This study investigates a single-stage volute-type centrifugal pump, performing unsteady numerical calculations of the full flow field based on BANS to explore the unsteady flow characteristics of the impeller, with particular emphasis on analyzing the pressure distribution characteristics at each node within the circumferential flow domain that varies along the axial direction at the impeller outlet. The results demonstrate that: intense turbulent fluctuations occur on the blade pressure surface and in the blade trailing-edge wake region; under various operating conditions, the pressure coefficient in the circumferential direction at the impeller outlet produces extreme values in the vicinity of the volute tongue; at the rated operating condition of Q/QN = 1, the pressure coefficient in the circumferential direction at the impeller outlet exhibits a stable periodic distribution, while with changing operating conditions, the non-uniformity of the pressure coefficient distribution along the volute circumferential direction increases; under low-flow-rate operating conditions, significant flow separation occurs within the impeller flow domain, generating large-scale separation vortices in the flow passages that disrupt the periodic distribution characteristics of the pressure coefficient in the circumferential direction at the impeller outlet; the pressure coefficient at the impeller outlet circumference is distributed nearly symmetrically in the flow domains on both sides of the mid-span section; under the low-flow-rate operating condition of Q/QN = 0.2, numerous complex excitation signals appear in the low-frequency range of the pressure fluctuation spectrum, which is believed to be associated with the separation vortex structures within the flow domain.

Full Text

Study on the Internal Flow Characteristics of Centrifugal Pumps under Non-design Conditions

GAO Bo, WANG Xing-Ning, LI Zhong, ZHANG Ning, YANG Min-Guan
School of Energy and Power Engineering, Jiangsu University, Zhenjiang 212013, China

Abstract

This study investigates a single-stage volute centrifugal pump using unsteady Reynolds-averaged Navier-Stokes (RANS) simulations to explore unsteady flow characteristics in the impeller, with particular emphasis on pressure distribution features at nodes within the circumferential plane at the impeller outlet along the axial direction. Results demonstrate that intense turbulence pulsation occurs on the blade pressure surface and in the trailing edge wake region. The pressure coefficient Cp in the circumferential direction at the impeller outlet exhibits extreme values near the tongue under various operating conditions. At the design condition (Q/QN=1), Cp displays a stable periodic distribution, but this periodicity degrades as operating conditions deviate from design, with increasing non-uniformity along the volute circumference. Under small flow conditions, significant flow separation occurs in the impeller domain, generating large-scale separation vortices that disrupt the periodic Cp distribution. The Cp distribution is nearly symmetric about the mid-span plane. At Q/QN=0.2, numerous complex excitation signals appear in the low-frequency range of the pressure pulsation spectrum, which is believed to be associated with separation vortex structures in the flow domain.

Keywords: Centrifugal pump; unsteady flow; unsteady pressure distribution; flow separation

---

Centrifugal pumps serve as fundamental energy conversion and fluid transport devices widely employed in industrial and agricultural applications, aerospace propulsion, and daily life. The interaction between the asymmetric volute and high-speed rotating impeller creates rotor-stator interference, leading to operational instability and time-dependent pressure pulsations that degrade pump performance, cause mechanical component fatigue, and induce vibration and noise. Consequently, detailed analysis of internal pressure distribution and pulsation characteristics has become a critical research focus.

Scholars worldwide have extensively investigated unstable flow in centrifugal pumps. Yuan et al. employed Large Eddy Simulation (LES) to analyze unsteady flow structures, attributing significant pressure pulsation variations to rotor-stator interaction. Wang et al. conducted numerical and experimental studies on pressure pulsation characteristics at the blade inlet and volute base circle using LES, finding that the pressure distribution period along the volute base circle matches the blade number. Nevertheless, due to the complexity of internal unsteady flow fields, the underlying mechanisms remain under computational and experimental investigation.

This study employs Reynolds-averaged Navier-Stokes (RANS) simulations with the ω-k turbulence model to perform three-dimensional unsteady numerical calculations of a single-stage volute centrifugal pump, focusing on unsteady flow characteristics at the impeller outlet. Statistical methods analyze pressure distribution features at nodes within the circumferential plane along the axial direction at the impeller outlet. Comparative analysis across different operating conditions reveals the unsteady flow characteristics in the impeller outlet region, providing theoretical support for research on unsteady flow in centrifugal pumps.

1.1 Model Pump Parameters

The basic performance parameters of the centrifugal pump used in this study are shown in Table 1 . The three-dimensional solid models of the impeller and volute were created using Pro/E5.0.

1.2.1 Mesh Generation and Computational Model

The entire model consists of a rotating impeller water body, a stationary volute water body, and inlet/outlet extension sections, with these four subdomains forming the numerical solution domain. ICEM meshing software was used to generate structured grids for the entire computational domain, with a total of 1,128,281 cells, including 566,716 in the impeller region and 435,128 in the volute region. Grid independence verification showed that when cell count exceeded 1.1 million, further increases caused minimal fluctuations in head and efficiency results. Therefore, the physical model and discretization scheme achieved spatial convergence and numerical stability, meeting accuracy requirements. The local grid distribution in the computational domain is shown in Figure 1 [Figure 1: see original paper]. To obtain more accurate results, local grid refinement was applied at radial cross-sections in flow regions such as around blades and volute tongue, adjusting node density accordingly.

1.2.2 Turbulence Model Selection

To investigate turbulence model effects, standard k- , ω-k, and SST models were used to calculate pump energy performance under identical boundary conditions, with results compared against experimental data. Figure 2 [Figure 2: see original paper] compares calculated and experimental performance curves under various conditions, where $=gH/u^2$ represents the dimensionless head coefficient and the abscissa shows flow ratio Q/QN. Comparison reveals that all three turbulence models produced energy performance curves generally consistent with experimental trends. For Q/QN=0.6–1.2, calculated head values agreed well with experiments, with maximum error <5%. At high flow rates, numerical error increased with flow, but results among models differed little. Efficiency curves show that the ω-k model provided the best agreement with experiments at small flow rates. Considering the need for accurate external characteristic prediction and accounting for convergence and error factors, the standard ω-k model was selected.

2.2.3 Boundary Conditions

Unsteady flow calculations were performed using ANSYS CFX13.0 with multiple reference frames. Walls used no-slip boundary conditions with standard wall functions for near-wall regions. Considering manufacturing limitations, wall roughness was set to 0.1 mm. Inlet used velocity boundary condition, outlet set as free outflow. For unsteady calculations, the impeller region moved relative to other grids, with rotor-stator interface set to transient rotor-stator mode, crucial for capturing rotor-stator interaction. Unsteady simulation used steady results as initial conditions, with time step ΔT=1.149$×10^{-4}$s per 1° impeller rotation, completing 360 steps per revolution.

3.1 Turbulent Kinetic Energy Distribution Characteristics

To analyze periodic turbulence pulsation intensity, turbulence intensity coefficient Tu was defined based on literature [9] as:

$$Tu = K/U_2^2$$

where K is turbulent kinetic energy and U_2 is impeller outlet velocity. Figure 3 [Figure 3: see original paper] shows Tu distribution at the impeller mid-span at a given moment for Q/QN=0.2 and 1.4 conditions, obtained by unwrapping the impeller domain. At Q/QN=0.2, turbulent kinetic energy changes dramatically near blade pressure surfaces and trailing edge wake regions, showing significant turbulence instability and intense pulsation with large energy dissipation. In contrast, at Q/QN=1.4, turbulent kinetic energy changes little due to absence of flow separation.

3.2 Pressure Distribution Characteristics at Impeller Outlet

Pressure distribution uses normalized form through dimensionless pressure coefficient Cp, defined as the ratio of pressure normalized by impeller circumferential velocity U_2. With P (x,y,z,t) representing pressure at any grid node, the dimensionless pressure coefficient [10] is:

$$Cp = (P - P_{ref})/(½ U_2^2)$$

To obtain pressure variation at impeller outlet circumferential nodes during rotation, unsteady full-domain calculations yielded pressure coefficient Cp distributions along the circumferential plane at varying axial positions at impeller outlet, shown in Figure 4 [Figure 4: see original paper]. During pump operation, Cp at impeller outlet circumference reaches extreme values near the tongue for all conditions (minimum points appear at =26°), related to strong rotor-stator interaction at the tongue. Compared with small flow conditions, rated condition Q/QN=1 shows stable periodic Cp distribution with period matching blade count and uniform pressure distribution at each passage outlet. Under small flow conditions, Cp shows increasing trend along circumferential direction when >26°, with Q/QN=0.2 showing particularly significant periodic variation changes and two peaks within one impeller passage. This results from complex flow and obvious flow separation in the impeller domain under small flow conditions, generating vortices that destroy flow stability and periodicity.

Figure 5 [Figure 5: see original paper] shows corresponding relative velocity distributions in the impeller domain. Under small flow conditions, large-scale flow separation vortices appear at blade inlet, blade outlet pressure side, and blade outlet suction side, with some passages showing severe blockage. At rated condition, relative velocity distribution is uniform.

Statistical analysis of pressure distribution along blade thickness variation at impeller outlet circumference yields Cp distribution at Q/QN=1.4 shown in Figure 6 [Figure 6: see original paper]. Large pressure gradients exist before and after the tongue. Contrary to small flow conditions, large flow condition shows Cp minimum before the tongue, an inflection point near the tongue, followed by Cp maximum, then decreasing along circumferential direction. Overall, Cp variation period is approximately 60°, consistent with blade passage count. Cp distribution is nearly symmetric about the mid-span plane.

3.3 Pressure Pulsation Spectrum Analysis at Volute Measurement Points

To analyze pressure pulsation characteristics in the volute caused by unstable internal flow, monitoring points were placed during unsteady calculations for time-domain signal acquisition, with Fast Fourier Transform yielding pressure pulsation frequency spectra shown in Figure 7 [Figure 7: see original paper]. Points P4 and P5 were selected to analyze off-design condition characteristics, where P4 represents volute Section IV and P5 represents volute Section II. Frequency spectra show frequency f on the abscissa and pulsation amplitude A on the ordinate.

With impeller rotational speed n=1450 r/min, shaft frequency f_R=24.2 Hz and blade number Z=6, blade passing frequency f_{BPF}=145 Hz. Spectra show pulsation amplitudes concentrated in low-frequency region, with main peak at blade passing frequency 145 Hz. First harmonic amplitude significantly exceeds higher harmonics, indicating blade frequency dominance, with peaks also appearing at integer multiples of blade frequency. Additionally, at Q/QN=0.2, measurement points show larger pressure pulsation amplitudes than at Q/QN=1.4, with more excitation frequency signals in low-frequency band, related to complex flow characteristics under small flow conditions and associated with separation vortex structures.

Conclusions

1) Pressure coefficient Cp at impeller outlet circumference shows extreme values near the tongue under different conditions. At rated condition, Cp distribution is stably periodic with uniform pressure at each passage outlet. As conditions change, Cp non-uniformity along volute circumference increases.
2) Small flow conditions cause obvious flow separation in the impeller domain, generating large-scale separation vortices that destroy the periodic Cp distribution characteristics at impeller outlet circumference.
3) Pressure coefficient Cp at impeller outlet shows large pressure gradients before and after the tongue, with nearly symmetric distribution about the mid-span plane.
4) At Q/QN=0.2, more complex excitation frequency signals appear in the low-frequency band, likely related to separation vortex structures in the flow domain.

References

[1] C E Brennen, Hydrodynamics of Pumps, Oxford University Press, 2011.
[2] Raul Barrio, Jorge Parrondo, Eduardo Blanco. Numerical Analysis of the Unsteady Flow in the Near-Tongue Region in a Volute-type Centrifugal Pump for different Operating Points[J]. Compute and Fluids, 2010, 39: 859-870.
[3] Q Z Liu, K Yang, D Y Liu eal. Research of fluid-induced pressure fluctuation due to impeller-volute interaction in a centrifugal pump[J]. Materials Science and Engineering 2013,52: 022026.
[4] Spence R, Amaral-Teixeira J. A CFD parametric study of geometrical variations on the pressure pulsations and performance characteristics of a centrifugal pump[J]. Compute Fluids, 38(2009) 1243-1257.
[5] R Dong, S Chu, and J Katz. Effect of modification to tongue impeller geometry on unsteady flow, pressure fluctuations, and noise in a centrifugal pump.Journal of Turbomachinery.vol. 119, no. 3, pp. 506–515, 1997.
[6] K A Kaupert and T Staubli. The unsteady pressure field in a high specific speed centrifugal pump impeller—part I: influence of the volute. Journal of Fluids Engineering.vol. 121, no. 3, pp. 621–626, 1999.
[7] 袁建平, 付燕霞, 刘阳等. 基于大涡模拟的离心泵蜗壳内压力脉动特性分析 [J]. 排灌机械工程学报. 2010, 28(04): 310-314
YUAN Jianping, FU Yanxia, LIU Yang, et al. Analysis on pressure fluctuation within volute of centrifugal pump based on large eddy simulation[J]. Journal of Drainage and Irrigation Machinery Engineering, 2010, 28(04): 310-314
[8] W J Wang, Y R Cui, Y Wang eal. Analysis on the blade inlet pressure

fluctuation of the centrifugal pump based on LES[J]. Materials Science and Engineering. 2013,52: 022025.
[9] Feng J, Benra F-K, Dohmen H. Investigation of periodically unsteady flow in a radial pump by CFD simulations and LDV measurements [J]. ASME Journal of Turbomachinery, 2011, 133(1) : 011004.
[10] Ji Pei, Shouqi Yuan, Friedrich-Karl Benraetl. Numerical Predication of Unsteady Pressure Field Within the Whole Flow Passage of a Radial Single-Blade Pump[J]. Journal of Fluids Engineering., 2012, Vol. 134: 101103

Note: Figure translations are in progress. See original paper for figures.