International Journal of Heat and Fluid Flow 72 (2018) 52–60

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

journal homepage: www.elsevier.com/locate/ijhff

Flow characteristics in a volute-type centrifugal pump using large eddy simulation

Beomjun Kye[a], Keuntae Park[a], Haecheon Choi[a,*], Myungsung Lee[b], Joo-Han Kim[b]

[a] Department of Mechanical and Aerospace Engineering, Seoul National University, 1, Gwanak-ro, Gwanak-gu, Seoul 08826, Republic of Korea
[b] Intelligent Mechatronics Research Center, Korea Electronics Technology Institute, 388, Songnae-daero, Bucheon 14502, Republic of Korea

ARTICLE INFO

Keywords:
Large eddy simulation
Centrifugal pump
Flow separation
Leakage
Impeller-volute interaction

ABSTRACT

The flow characteristics in a volute-type centrifugal pump operating at design (Q_d = 35 m^3/h) and off-design (Q_{off} = 20 m^3/h) conditions are investigated using large eddy simulation. Numerical results indicate that separation bubbles are generated on both the pressure and suction sides of impeller blades. At the off-design condition, the blade pressure side contains a larger recirculation zone with highly unsteady characteristics due to impeller-volute interactions. The vortices shed from a blade trailing edge due to its rotation strongly interact with those from the following blade and leakage through radial gaps at the off-design condition, generating stronger vortices in a wider region inside the volute, whereas this mutual interaction is weak at the design condition. Flow separation also occurs around the volute tongue at both operating conditions. At the off-design condition, a part of high-pressure fluid discharged from the volute does not follow the main stream to the outlet duct but re-enters into the volute area near the volute tongue. This pressurized fluid forms a high adverse pressure gradient on the blade pressure side, resulting in strong unsteady separation there. Also, a high pressure gradient in the axial direction at the radial gaps is formed especially near the volute tongue, creating the leakage into the cavities. Inside the volute, azimuthal vortices exist and grow along the volute passage. A secondary motion induced by these vortices also significantly affects the leakage to the cavities. All of these flow losses contain unsteady features that are strongly influenced by impeller-volute interactions, especially at the off-design condition.

1. Introduction

A centrifugal pump, which is one of the most commonly used turbomachines, is widely utilized in residential buildings as well as in industries. In centrifugal pumps, complex three-dimensional flow phenomena involving turbulence, secondary flows and unsteadiness occur (Brennen, 1994). To meet various ranges of pressure rise and flow rates required, many centrifugal pumps are operated at off-design conditions as well as at the design condition. At off-design conditions, the flow characteristics inside centrifugal pumps become more complex than those at the design condition. Commonly used steady Reynolds-averaged Navier-Stokes (RANS) turbulence models often inaccurately predict the flow inside centrifugal pumps at off-design conditions (Byskov et al., 2003). For this reason, a large eddy simulation (LES) technique is a promising alternative to predict such complex flow phenomena inside centrifugal pumps.

Kato et al. (2003) conducted LES to predict the flow in a mixed-flow pump at off-design conditions, where the standard Smagorinsky model together with the van Driest damping function near the wall was used as a subgrid-scale model. The results from LES were compared with those measured by a laser Doppler velocimetry (LDV) and good agreements were obtained. Byskov et al. (2003) studied the flow in a shrouded six-bladed centrifugal pump impeller at design and off-design conditions using the localized dynamic Smagorinsky model (Piomelli and Liu, 1995). They showed that, at an off-design condition, LES predicts complex flow phenomena, such as steady nonrotating stalls and flow asymmetry between impeller passages, better than RANS. Although most of previous LESs of flows in centrifugal pumps used the standard Smagorinsky model or simulated flow only in rotating parts, LESs using more advanced subgrid-scale models such as a filtered structured function model (Posa et al., 2011) or a wall-adapting local eddy-viscosity model (Posa et al., 2015, 2016) were also performed to study the flow structures in a mixed-flow pump including all rotating and stationary parts. They showed that LES accurately predicts unsteady flow features, such as separated flow near the blade surfaces, backward flow near the shroud surface and rotor-stator interaction, at design and off-design conditions. A detached eddy simulation (DES) was also conducted to study the flow in a radial diffuser pump at low

* Corresponding author at: Institute of Advanced Machines and Design, Seoul National University, Republic of Korea.
E-mail address: choi@snu.ac.kr (H. Choi).

https://doi.org/10.1016/j.ijheatfluidflow.2018.04.016
Received 5 December 2017; Received in revised form 30 March 2018; Accepted 30 April 2018

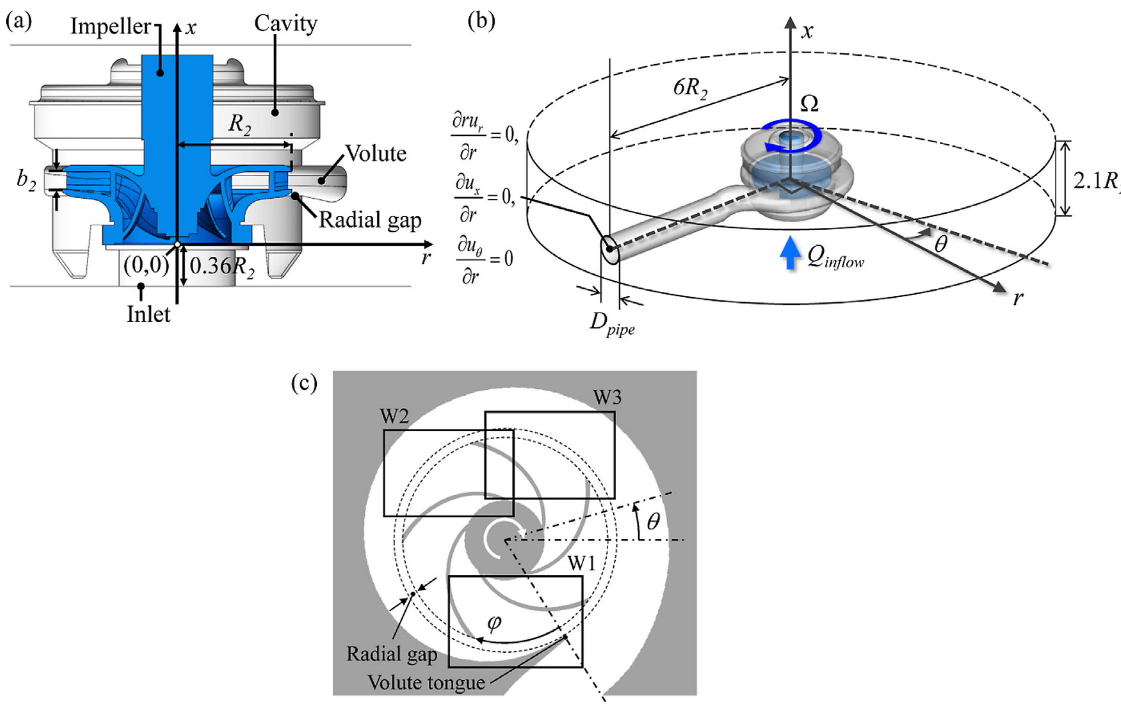

Fig. 1. Schematic diagram of the coordinates, computational domain, and boundary conditions: (a) two-dimensional view on an x–r plane; (b) perspective view; (c) plan view of the impeller and volute casing. In (c), φ is the blade position angle, and the windows, W1–W3, are specific flow areas of our interest at $\varphi = 46.8^{\circ}$.

flow rates (Feng et al., 2009). The results from DES showed good agreements with those from particle image velocimetry experiments, capturing flow separation and back flows inside the impeller.

Although a number of successful LESs have been conducted to predict flow inside centrifugal pumps, LESs of the flow in volute-type centrifugal pumps including both the impeller and volute casing have been fairly limited. Instead, RANS has been employed to investigate the impeller-volute interaction (Gonzalez et al., 2002; Asuaje et al., 2005; Barrio et al., 2010; Cheah et al., 2011) and to improve the volute-type pump efficiency by optimizing the geometries of the impeller and volute (Barrio et al., 2008; Spence and Amaral-Teixeira, 2009; Alemi et al., 2015). Volute-type centrifugal pumps, which operate without diffuser vanes, show complex flow features near the tongue due to direct interactions between the impeller and volute tongue. Recently, Zhang et al. (2016) performed an LES using a commercial software to analyze the flow in a volute-type centrifugal pump, where the standard Smagorinsky model was used as a subgrid-scale model. They reported that the impeller-volute interaction is dominated by vortex shedding in the wake of the blade trailing edge and pressure pulsations are strongly associated with the corresponding vorticity magnitude inside the pump. However, detailed flow features such as flow separation, leakage flow and impeller-volute interaction inside the volute-type centrifugal pump have not been fully understood. Therefore, in the present study, we conduct an LES of flow in a volute-type centrifugal pump including both the impeller and volute casing to understand the flow characteristics. A dynamic global model is used as a subgrid-scale model (Park et al., 2006; Lee et al., 2010), and an immersed boundary (IB) method in a non-inertial reference frame (Kim and Choi, 2006) is used to satisfy the no-slip condition at the stationary and rotating solid surfaces. The pump operates at the Reynolds number of $Re = 1{,}700{,}000$ based on the radius of the impeller blade R_2 and the blade tip velocity U_2.

2. Numerical details

The governing equations for LES are spatially filtered Navier-Stokes and continuity equations in a non-inertial reference frame:

$$\frac{\partial \overline{u}_i}{\partial x_i} - q = 0,$$
$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\overline{u}_j\overline{u}_i - w_j\overline{u}_i + \overline{u}_j w_i) = -\frac{\partial \overline{p}}{\partial x_i} + \frac{1}{\mathrm{Re}}\frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} + f_i, \quad (2.1)$$

where $u_i = v_i + \varepsilon_{ijk}\Omega_j x_k, w_i = \varepsilon_{ijk}\Omega_j x_k, x_i$ and v_i are the cylindrical coordinates and corresponding velocity vectors in the non-inertial reference frame, respectively, ε_{ijk} is the permutation tensor, Ω_j represents the angular velocity of the impeller, p is the pressure, f_i and q are the momentum forcing and mass source/sink to satisfy the no-slip condition and mass conservation on the immersed boundary (Kim and Choi, 2006), respectively, and Re is the Reynolds number. The overbar $\overline{(\cdot)}$ denotes a spatial filtering using a box filter and $\tau_{ij} \approx \overline{u_i u_j} - \overline{u}_i\overline{u}_j$ (Kumar and Mahesh, 2017) is the subgrid-scale stress tensor modeled by a dynamic global model (Park et al., 2006; Lee et al., 2010). The governing equations in the non-inertial reference frame (2.1) are written in a strongly conservative form (Beddhu et al., 1996; Kim and Choi, 2006), and all the variables are normalized by the impeller radius R_2 and the blade tip velocity U_2 (see Fig. 1(a)). Here, we adopt a non-inertial reference frame which is fixed to the rotating impeller. Then, the grids on and near the rotating impeller are fixed in time, and thus we can more accurately resolve the boundary layer flow near the impeller blades and shroud. However, due to the choice of this reference frame, the volute casing and outlet duct rotate in the opposite direction of the impeller rotation, and grid locations for momentum forcing and mass source/sink near those surfaces should be computed at each computational time step (note that the grids do not change in time owing to the use of immersed boundary method). A second-order semi-implicit fractional step method in a cylindrical coordinate is used to solve (2.1). At $r \le 0.72R_2$, the convective and diffusive terms with azimuthal derivatives are treated implicitly, but the axial and radial derivative terms are treated explicitly. At $r > 0.72R_2$, the convective and diffusive terms with radial derivatives are treated implicitly, while the axial and azimuthal derivative terms are treated explicitly. A third-order Runge-Kutta (RK3) and Crank-Nicolson methods are used for the explicit and implicit terms, respectively. Further details are found in Akselvoll and Moin (1996). For spatial derivative terms, a third-order QUICK scheme is applied only at inlet region ($x/R_2 < 0$), and the