Springer

KSME

Journal of Mechanical Science and Technology 27 (6) (2013) 1643~1648
www.springerlink.com/content/1738-494x
DOI 10.1007/s12206-013-0411-5

Comparison of cavitation prediction for a centrifugal pump with or without volute casing †

Xianwu Luo[1,2,*], Wei Wei[1], Bin Ji[1], Zaibing Pan[3], Wenchao Zhou[3] and Hongyuan Xu[1]

[1]*State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing, 100084, China*
[2]*Beijing Key Laboratory of CO_2 Utilization and Reduction Technology, Tsinghua University, Beijing, 100084, China*
[3]*KSB Shanghai Pump Co., Ltd, 1400 Jiangchuan Road, Minhang, Shanghai, 200245, China*

(Manuscript Received November 23, 2012; Revised February 27, 2013; Accepted March 4, 2013)

Abstract

Cavitation may not only cause head and efficiency breakdown of hydraulic machines but also generate other unfavorable phenomena such as noise and vibration. Therefore, the accurate prediction of cavitation development is important for various pump applications. In this paper, two numerical models, namely, models A and B, are applied to simulate the turbulent cavitating flows inside a centrifugal pump to investigate the effect of calculation domain on the prediction accuracy of cavitation performance for hydraulic machines. Model A has a calculation domain with volute casing, whereas model B has a single blade-to-blade flow passage without volute casing. Steady simulations of cavitating flow in the pump have been conducted based on the shear stress transport k-ω turbulence model and the homogeneous cavitation model. Both models A and B predicted that the pump performance decreases with decreasing cavitation number. Experimental results show that model B can predict better the critical cavitation number at the best efficiency point compared with model A, which is the full flow passage model. Internal flow investigations indicate that an asymmetrical feature of cavitating flow exists when the calculation domain with volute casing is applied. The asymmetrical cavitation development in different blade-to-blade flow passages for model A results in an over-estimation of the decrease in pump performance because of the interaction between the impeller blade and the tongue of the volute casing. A simple calculation domain without volute casing is preferred for steady cavitation prediction in pumps rather than the full flow passage with volute casing because the former has better convergence, less resource requirements, and lower time consumption.

Keywords: Centrifugal pump; Volute casing; Numerical simulation; Hydraulic performance; Cavitating flow

1. Introduction

Cavitation, which is an important physical phenomenon in the operation of hydraulic machines, may result in many undesirable effects such as head and efficiency drop, noise, and vibration [1, 2]. However, cavitating flows in various applications, particularly in hydraulic machines, are complicated. The mechanism of cavitation flows remains unclear; therefore, although cavitation is an old topic, the behavior of cavitation still attracts much attention from the engineering community.

In recent years, cavitation in hydraulic machines has been extensively studied [3, 4]. Most studies mainly focused on the (1) numerical methods of cavitation prediction [5, 6] and (2) performance improvement through the application of cavitating flow simulation [7, 8]. These contributions have prompted the need for explanations regarding cavitation in hydraulic machines. Studies show that turbulent cavitating flow simulation predicts the cavitation development in flow passages, and the flow upstream of the impeller inlet is crucial for cavitation performance [9, 10].

However, studies focusing on simulating the cavitating flow by using the cavitation model and the turbulence model are still limited. Numerical simulations of cavitating flow in hydraulic machines require adequate computer resource and consume much time. Given that cavitation analysis improves the performance and reliability of hydraulic systems, a suitable numerical method with high accuracy is necessary considered for cavitation prediction.

Simulating cavitation in hydraulic machines by using full flow passage is possible with the improvement of computing technology. Applying full flow passage and unsteady simulation is necessary to analyze the cavitation in machines because cavitation in hydraulic machines, which often have rotating parts, is unsteady and inhomogeneous. However, unsteady simulation with full flow passage cannot meet the requirement of short product development cycle because numerical models

*Corresponding author. Tel.: +86 10 62789853, Fax.: +86 10 62785285
E-mail address: luoxw@tsinghua.edu.cn
† This paper was presented at the ISFMFE 2012, Jeju, Korea, October 2012. Recommended by Guest Editor Hyung Hee Cho

Table 1. Geometrical parameters of the test pump.

Parameters	Symbols	Values
Casing shape	—	volute
Blade number	Z	5
Blade exit angle	β_2 / (°)	25
Blade width ratio at the impeller exit	b_2/D_2	0.0666
Blade inlet diameter ratio at the tip	D_{1t}/D_2	0.618
Blade inlet diameter ratio at the hub	D_{1h}/D_2	0.414
Impeller inlet diameter ratio	D_0/D_2	0.616
Hub ratio	d_h/D_2	0.200

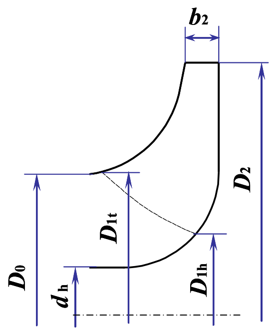

Fig. 1. Meridional configuration of the test pump impeller.

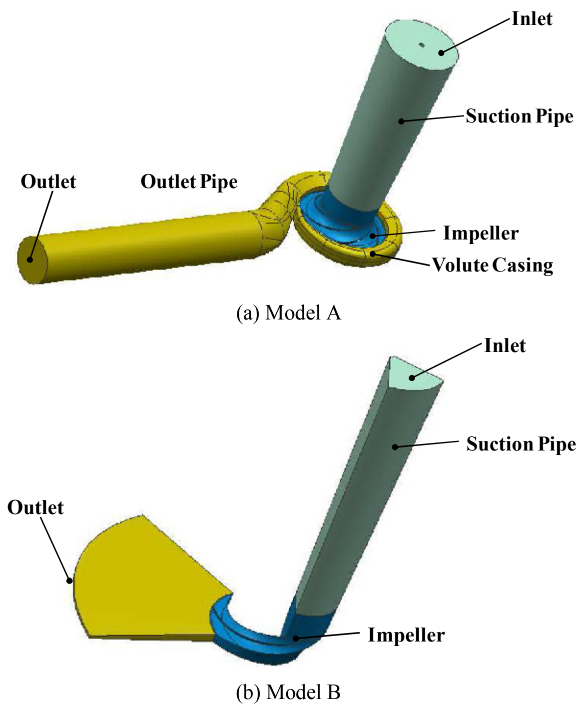

Fig. 2. Calculation domains for models A and B.

are resource consuming and are not suitable for designing hydraulic machines. The steady simulation with single blade-to-blade flow passage is preferred in engineering applications.

This paper compares the cavitation prediction in a centrifugal pump by using different calculation domains with or without volute casing. On the basis of the numerical simulation and experiments, the effects of calculation domain on the prediction of pump cavitation performance are discussed in detail.

2. Geometry for test pump

The test pump in this paper is designed to have the following operating parameters: Q_d = 5.903 m^3·min^{-1} at the rotational speed n = 1480 min^{-1}. The specific speed of the pump n_s is 226 m·m^{3}min^{-1}·min^{-1}. Although the specific speed of this pump is popular, its cavitation performance is strictly requested because of its application as condensate pumps at nuclear power plants.

The meridional configuration for the impeller is shown in Fig. 1, and the geometrical parameters of the test pump are listed in Table 1. To achieve good cavitation performance, the impeller diameter ratio between the blade leading edge and the trailing edge, as well as the section area at the blade inlet, uses large values [8].

An appropriate volute casing that matches the impeller is designed for the investigation of the effect of calculation domain with or without volute casing on the prediction accuracy of cavitation performance. The volute casing is conventional. The basic configuration can be seen in Fig. 2(a).

3. Method descriptions

3.1 Numerical methods

3.1.1 Simulation models and grid generations

Two 3D numerical models, namely, models A and B, are utilized to investigate the influence of the volute casing on the cavitation simulation for centrifugal pumps. In particular, model A, which has a calculation domain of a full flow passage, includes the suction pipe, impeller, volute casing, and outlet pipe. By contrast, model B, which does not have a volute casing, is a single blade-to-blade flow passage model. The two models are shown in Figs. 2(a) and 2(b), respectively. Note that a flow zone with an identical section area and the downstream of the impeller exit are added for model B for better calculation convergence. Model B is clearly simpler compared with model A.

The structured hexahedral mesh is applied for the suction pipe, outlet pipe, and impeller to obtain accurate simulation results. The grids are well refined at the area around the interfaces between the different flow components. The same topological structure and mesh point distribution are applied for the impeller flow passages of both models. An unstructured tetrahedral mesh is generated for the volute casing because the volute casing has a complex structure and has 1740060 elements. To obtain better balance near the interfaces, the mesh generation for the volute casing was given more elements than the impeller and other flow components. Model A has 3130465 elements, which is approximately five times of that for model B. This difference denotes that the domain with the volute casing, i.e., model A, will consume a larger calculation

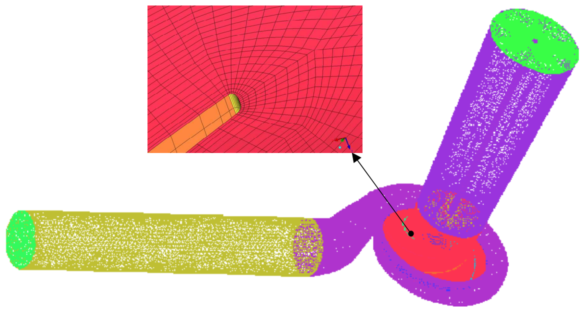

Fig. 3. Mesh generation for model A.

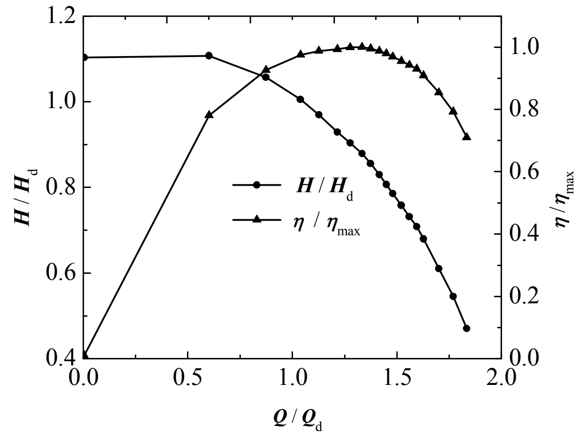

Fig. 4. Performance curves for the test pump (n = 1480 min^{-1}).

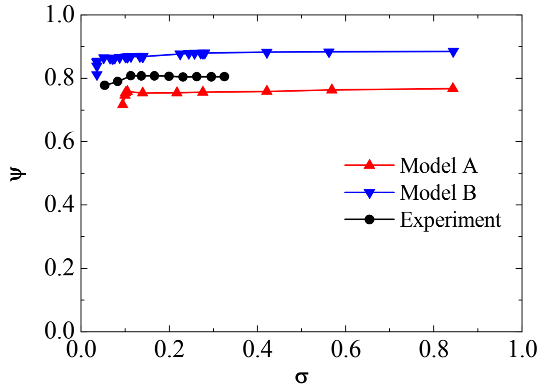

Fig. 5. Cavitation performance for the test pump at BEP (n = 1480 min^{-1}).

resource than that of model B. Fig. 3 shows the grid generated for model A. The mesh quality for both models is good.

3.1.2 Boundary conditions

Calculations are conducted based on Reynolds averaged Navier–Stokes equations using the commercial code ANSYS CFX. The shear stress transport k-ω turbulence model [11] and the homogeneous cavitation model [12] are applied for turbulent cavitating flow simulation. The fluids used in the simulation are water and vapor at 25°C.

The boundary conditions are as follows:

(1) The inlet of the calculation domain is specified by using the averaged mass flow-rate based on the mass equilibrium. One-fifth of the total mass flow is set because model B merely includes a single blade-to-blade flow channel.

(2) At the outlet plane, an averaged static pressure is set according to the total pressure level at the domain inlet. The set value is reduced gradually during cavitation development.

(3) All solid walls are set as the non-slip wall condition.

(4) The flow passage of the impeller for both models is set at rotating coordinate system, whereas the rest of the flow passages are set at a stationary reference frame. The interfaces between the two frames are treated by using the general grid interface method.

3.2 Experimental test

Both hydraulic and cavitation performances of the pump were tested experimentally to provide a reference to the simulation results. The experiment was conducted by using a close-loop test stand, which satisfies the standard of IEC60193-1999. The following sensors or gauges were used for the test:

(1) A torque meter was applied to measure the shaft torque and rotational speed.

(2) Pressure sensors were installed at the inlet pipe and outlet pipe of the test pump.

(3) A magnetic flow meter was set between the pump outlet pipe and the settling tank.

The hydraulic performance of the test pump at n = 1480 min^{-1} is illustrated in Fig. 4. The best efficiency point (BEP) has a larger flow discharge compared with the design point.

The cavitation performances at several operation conditions near the design point were tested by gradually decreasing the static pressure before the pump suction pipe. The experimental result of the cavitation performance at BEP is shown in Fig. 5.

4. Results and considerations

Three parameters are defined for comparing the experimental and calculation results for different calculation domains:

(1) Head coefficient ψ

$$\psi = H \Big/ \left(\frac{u_2^2}{2g}\right). \tag{1}$$

(2) Thoma's cavitation number σ

$$\sigma = NPSH / H . \tag{2}$$

(3) Cavitation specific speed S

$$S = n\sqrt{Q}\left(NPSH_r\right)^{-0.75}. \tag{3}$$

Steady calculations are conducted for both models A and B at the flow discharge Q = 8.02 m^3·min^{-1}, which is the BEP in the experimental test. Fig. 5 shows that the head coefficient,

Table 2. Performance comparisons for the test pump at BEP.

	$\psi/(\psi)_{exp}$	$\eta/(\eta)_{exp}$	σ_c
Experimental data	1.000	1.000	0.062
Calculation data via model A	0.955	1.098	0.100
Calculation data via model B	1.100	1.236	0.043

i.e., ψ, is a vertical axis and that Thoma's cavitation number, i.e., σ, is a horizontal axis. Fig. 5 also shows the cavitation performance curves for the two models and the experimental result.

The critical value of the required net positive suction head (*NPSHr*)c is defined at 3% head breakdown point. On the basis of the experimental result, the cavitation specific speed of the test pump at the best efficiency point is 2493 $m \cdot m^3 min^{-1} \cdot min^{-1}$. Table 2 compares both experimental and numerical hydraulic and cavitation performances. The following observations are noted on the basis of the aforementioned results:

(1) The hydraulic performance of the test pump is predicted fairly well, although a discrepancy of pump head exists between the experimental and numerical results. The differences resulted from the different outlet definitions. In the numerical results, model A has a longer outlet pipe, and model B has no volute casing.

(2) Both results of the two models show high efficiency because mechanical losses are not considered.

(3) The breakdown of head coefficient at low net positive suction head is reasonably predicted. The head coefficient drop for model A occurs earlier than that for model B.

(4) On the basis of the critical Thoma's cavitation number, i.e., σ_c, the prediction accuracy of model B is better than that of model A. The reason for this finding will be explained later.

(5) The calculation for model A overestimates the effect of cavitation development on pump performance.

To have a clear insight on the cavitation development in the flow passage, the cavity distributions and static pressure in the pump at BEP are shown in Figs. 7 and 8. The blades are numbered for convenience of flow analysis, as shown in Fig. 6. Note that the trailing edge of blade 1 is located near the tongue of the volute casing.

In Fig. 7, the cavity is shown by using the iso-surface for a vapor volume fraction of 0.1. For models A and B, the cavities develop with decreasing total pressure at the domain inlet. The cavitation inception occurs near the blade suction side at the leading edge and grows along the downstream direction of the blade-to-blade flow channel. The cavity distribution for both models are different even at large cavitation numbers such as $\sigma = 0.275$. With the decrease of cavitation number, the difference between the two models enlarges, and a large cavity near the suction side of blade 1 appears. This cavity extends near the tongue of the volute casing at $\sigma = 0.104$. The asymmetrical

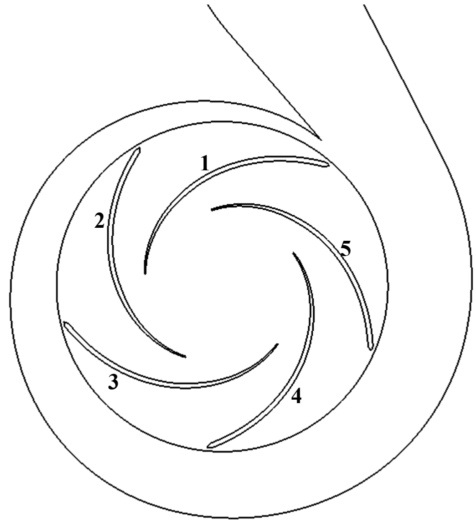

Fig. 6. Relative position of impeller blades.

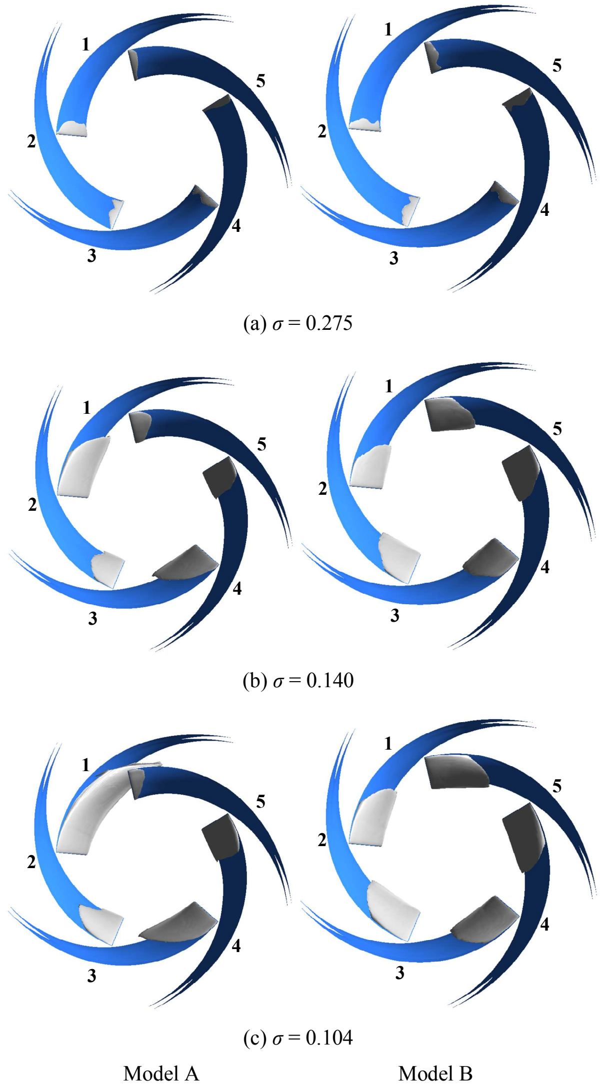

Fig. 7. Cavities in the pump impeller at BEP ($n = 1480\ min^{-1}$).

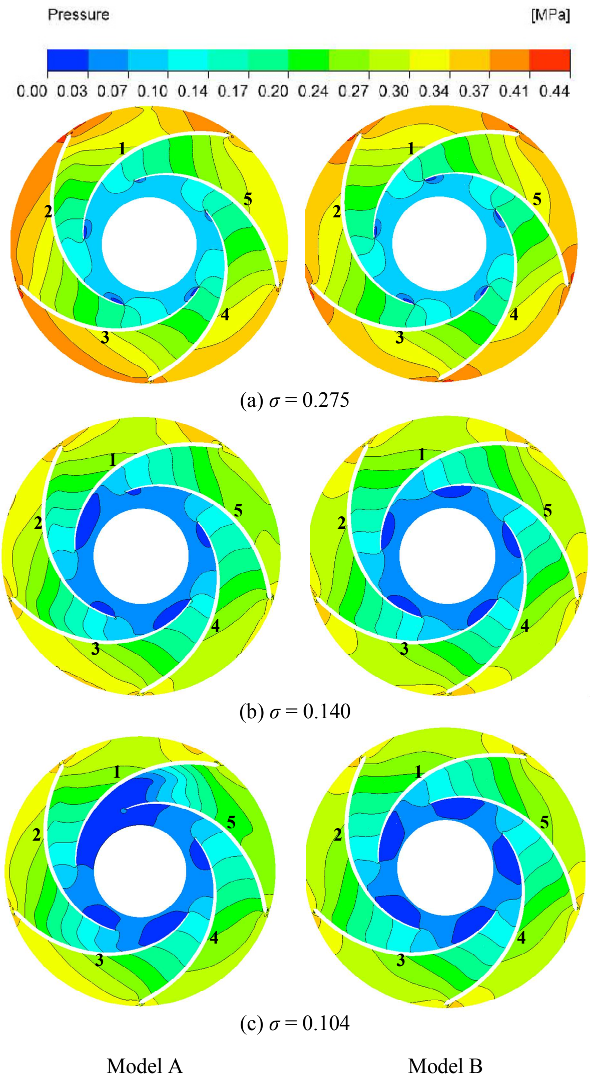

Fig. 8. Pressure distribution inside the impeller at BEP (n = 1480 min^{-1}).

feature of cavity distribution predicted by using model A is remarkable, as shown in Figs. 7(b) and 7(c). Compared with the results in Ref. [6], this feature is reasonable because the interaction between the impeller blades and the volute casing, particularly the casing tongue, greatly influences the cavity growth for steady calculation by using the full flow passage.

The cavity between blades 1 and 5 for model A has almost twice the length compared with that for model B. The cavity between blades 1 and 5 for model A is longer than one-half of the blade length and tends to reach blade 5 at $\sigma = 0.104$. This large cavity will cause severe flow blockage in the impeller and will result in a sudden drop of pump performance, as shown in Fig. 5. The asymmetrical feature of cavity distribution greatly influences the cavity growth in the flow passage between blades 1 and 5. Thus, the numerical prediction for model A overestimates the cavitation development compared with model B.

Fig. 8 shows the static pressure distribution on the mid-span plane of the pump impeller. The pressure increases along the blade from the leading edge to the trailing edge. The low-pressure area near the blade leading edge increases with decreasing cavitation number. An asymmetry also exists in the static pressure distribution for model A at small cavitation numbers. This asymmetry is decided by the unique cavity distribution of model A and resulted from the interaction between the blade and the casing tongue.

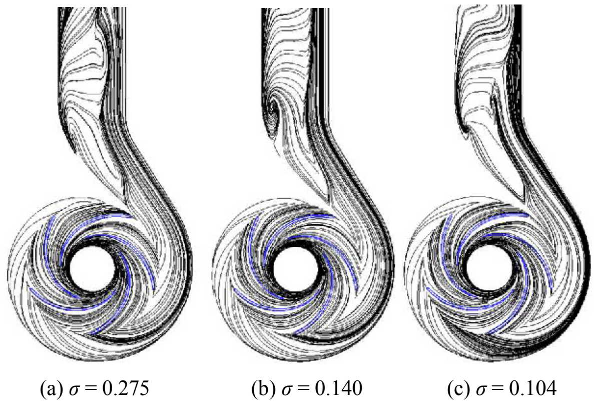

Fig. 9. Streak lines inside the pump at various cavitation conditions at BEP (n = 1480 min^{-1}).

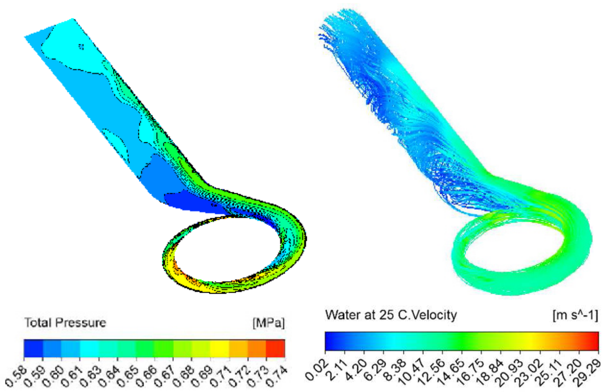

Fig. 10. Flow in the pump casing and outlet pipe at $\sigma = 0.104$ at BEP (n = 1480 min^{-1}).

The streak lines inside the pump at various cavitation conditions are shown in Fig. 9, and the flow in the pump casing and outlet pipe at $\sigma = 0.104$ is shown in Fig. 10. The flow distortion in the outlet pipe becomes more intense with decreasing cavitation number for model A and results in a large hydraulic loss.

For engineering applications, cavitation prediction should be conducted within a limited duration at the design stage. Steady simulation that applies single blade-to-blade flow passage as calculation domain can predict reasonable cavitation development in pumps, and the accuracy of this approach is acceptable in most cases [13]. A calculation domain without volute casing is more suitable for steady cavitation prediction in pumps compared with full flow passage with volute casing. In particular, the simple calculation domain is more preferable than complex calculation domains because the former has better convergence, less resource requirements, and lower time consumption compared with the latter.

5. Conclusions

(1) A remarkable asymmetrical feature of turbulent cavitating flow exists in the centrifugal pump when steady simulation is conducted by using the calculation domain with volute casing.

(2) The asymmetrical feature of cavitation development in the pump is caused by the interaction between the impeller blade and the volute casing tongue.

(3) Steady calculation overestimates cavitation development using the calculation domain with volute casing.

(4) For engineering applications, the calculation domain with single blade-to-blade flow passage is preferable for steady cavitation prediction in centrifugal pumps.

Acknowledgment

This work was supported by the National Natural Science Foundation of China (Grant Nos. 51179091 and 51206087), Tsinghua University (Grant No. 20111081106), Tsinghua-Yuyuan Medical Fund (Grant No. 20240000512), and the Ministry of Science and Technology of China (Grant No. 2008KR0441).

Nomenclature

b_2 : Blade width at the impeller exit
D_2 : Diameter at the impeller exit
d_h : Diameter of the impeller hub
g : Gravitational acceleration
H : Head defined by the total pressure difference between the domain outlet and inlet, as shown in Fig. 2
n : Rotational speed
$NPSH$: Net positive suction head, $= (p_0\text{-}p_v)/(\rho g)$
$NPSH_r$: Required net positive suction head
n_s : Specific speed, $= (nQ^{0.5})/H^{0.75}$
p_0 : Total pressure at the inlet of calculation domain
p_v : Vapor pressure at 25°C, = 3169Pa
Q : Flow discharge
Q_d : Flow discharge at the design point
S : Cavitation specific speed
u_2 : Peripheral speed at the impeller exit, $= \omega D_2/2$
σ : Blade angle
ρ : Efficiency
η : Density of a fluid
β : Thoma's cavitation number
ω : Angular speed, $= 2\pi n/60$

References

[1] X. W. Luo, B. Ji and H. Y. Xu, *Design and optimization of fluid machinery*, Tsinghua University Press, Beijing, China (2012).

[2] X. W. Luo, B. Ji, X. X. Peng, H. Y. Xu and M. Nishi, Numerical simulation of cavity shedding from a three–dimensional twisted hydrofoil and induced pressure fluctuation by Large-Eddy Simulation, *Trans. ASME J. Fluids Eng.*, 134 (4) (2012) 041202 1-10.

[3] B. Ji, X. W. Luo, Y. L. Wu, X. X. Peng and H. Y. Xu, Partially-Averaged Navier–Stokes method with modified k–ε model for cavitating flow around a marine propeller in a non-uniform wake. *International Journal of Heat and Mass Transfer*, 55 (23-24) (2012) 6582-6588.

[4] W. Wei, X. W. Luo, B. Ji, B. T. Zhuang and H. Y. Xu, Cavitating flow investigation inside centrifugal impellers for a condensate pump, IOP Conf. Ser.: Earth Environ. Sci., 15 (2012) 032061 1-8.

[5] B. Ji, X. W. Luo, X. X. Peng, Y. L. Wu and H. Y. Xu, Numerical analysis of cavitation evolution and excited pressure fluctuation around a propeller in non-uniform wake. *International Journal of Multiphase Flow*, 43 (2012) 13-21.

[6] H. Ding, F. C. Visser and Y. Jiang, Demonstration and validation of a 3D CFD simulation tool predicting pump performance and cavitation for industrial applications, *Trans. ASME J. Fluids Eng.*, 133 (1) (2011) 011101 1-6.

[7] X. W. Luo, Y. Zhang, J. Q. Peng, H. Y. Xu and W. P. Yu, Impeller inlet geometry effect on performance improvement for centrifugal pumps, *Journal of Mechanical Science and Technology*, 22 (2008) 1971-1976.

[8] B. T. Zhuang, X. W. Luo, W. P. Yu, L. Zhu, Y. N. Li and H. Y. Xu, Design optimization for suction structure of multistage vertical barrel pump, *Journal of Drainage and Irrigation Machinery Engineering*, 28 (1) (2010) 1-6.

[9] X. W. Luo, S. H. Liu, Y. Zhang and H. Y. Xu, Cavitation in semi-open centrifugal impellers for a miniature pump. *Frontier of Energy and Power Engineering in China*, 2 (1) (2008) 31-35.

[10] Y. Zhang, X. W. Luo, Y. C. Yi, B. T. Zhuang and H. Y. Xu, Investigation on the flow field upstream of a centrifugal pump impeller, *Journal of Engineering Thermophysics*, 32 (S1) (2011) 33-36.

[11] F. R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA Journal*, 32 (8) (1994) 1598-1605.

[12] A. K. Singhal, M. M. Athavale, H. Y. Li and Y. Jiang, Mathematical basis and validation of the full cavitation model, *Trans. ASME J. Fluids Eng.*, 124 (3) (2002) 617-624.

[13] X. Luo, *A study on impeller inlet geometry suitable for a mini pump*, Doctoral dissertation of Kyushu Institute of Technology, Kitakyushu, Japan (2004).

Xianwu Luo received his B.S. and M.S. degrees from Tsinghua University, Beijing, China in 1991 and 1997 and his Ph.D. in Mechanical Engineering from Kyushu Institute of Technology, Japan, in 2004. He is currently an associate professor at Department of Thermal Engineering, Tsinghua University, China.