

CFD simulation of reverse water-hammer induced by collapse of draft-tube cavity in a model pump-turbine during runaway process

Xiaoxi Zhang, Yongguang Cheng¹, Linsheng Xia and Jiandong Yang

State Key Laboratory of Water Resources and Hydropower Engineering Science,
Wuhan University, Wuhan 430072, China

E-mail: zhangxiaoxi@whu.edu.cn

Abstract: This paper reports the preliminary progress in the CFD simulation of the reverse water-hammer induced by the collapse of a draft-tube cavity in a model pump-turbine during the runaway process. Firstly, the Fluent customized 1D-3D coupling model for hydraulic transients and the Schnerr & Sauer cavitation model for cavity development are introduced. Then, the methods are validated by simulating the benchmark reverse water-hammer in a long pipe caused by a valve instant closure. The simulated head history at the valve agrees well with the measured data in literature. After that, the more complicated reverse water-hammer in the draft-tube of a runaway model pump-turbine, which is installed in a model pumped-storage power plant, is simulated. The dynamic processes of a vapor cavity, from generation, expansion, shrink to collapse, are shown. After the cavity collapsed, a sudden increase of pressure can be evidently observed. The process is featured by a locally expanding and collapsing vapor cavity that is around the runner cone, which is different from the conventional recognition of violent water-column separation. This work reveals the possibility for simulating the reverse water-hammer phenomenon in turbines by 3D CFD.

1. Introduction

Vapor cavities develop when the local water pressure drops to the vapor pressure in hydraulic systems during hydraulic transients. The cavities expand and shrink dynamically, accompanied by water-hammer wave propagation and reflection in the system. If the local pressure changes are too violent, water-column separation and collapse may occur. Moreover, the collapse of vapor cavities or the rejoining of the separated water columns can lead to large and nearly instantaneous surges in pressure, which is normally called as the reverse water-hammer. This phenomenon generates severe load on hydraulic structures and has potentials of causing the lift-up of turbine-generator units. Therefore,

¹ To whom any correspondence should be addressed, E-mail: ygcheng@whu.edu.cn



accurate predictions of occurrence possibility and reverse water-hammer values are very important for designing new projects.

Traditionally, one-dimensional (1D) numerical approach is considered an effective way for reverse water-hammer simulation. Bergant and Simpson et al.[1] showed that the mathematic models and numerical methods for simulating such cavitation transient flow have been relatively mature, and there are many standard 1D commercial codes for engineering applications in pipeline systems. For more complex hydropower systems, Nicolet et al.[2-4] studied the reverse water-hammer in a Francis pump-turbine draft-tube during emergency shutdown in generating mode, using 1D transient simulation code SIMSEN. The results showed that the 1D method was able to capture the sudden pressure rise induced by the vapor cavity collapse at the draft-tube inlet. However, the reverse water-hammer pressure in the draft-tube of the pump-turbine obtained by the 1D approach was not satisfactory. Firstly, the influences of cavitation on pump-turbine characteristics are not considered in the existing approach. Secondly, the 1D approaches are not able to capture the instantaneous minimum pressure caused by the interaction between the rotor and stator[5]. More importantly, the flow patterns in a draft-tube, especially downstream of the impeller, are asymmetrical, which is not in accordance with the basic hypotheses for 1D approach.

Nowadays, three-dimensional Computational Fluid Dynamics (3D CFD) has been used to study the complex flow patterns in hydro-turbines, and the Rayleigh-Plesset equation incorporated CFD is proved to be capable of modeling the formation and decay of vapor cavities. Liu et al.[6] analyzed the cavitating flow in a model Francis turbine, in which the cavitation characteristics and the overload cavitating vortex rope at the draft-tube were reproduced reasonably. Jošt and Lipej[7] showed that the pressure pulsations caused by the cavitating vortex rope can be predicted accurately both in frequency and amplitude. Kumar et al.[8] illustrated that the 3D CFD has great potential in analyzing the cavitation in hydro-turbines. However, there is no report available about using 3D CFD for simulating cavitating flow during transient processes, especially considering the reverse water-hammer.

In this study, we try to simulate the reverse water-hammer in the draft-tube of a pump-turbine during the runaway process in a model pumped storage system by 3D CFD. The aim is to explore reasonable approach for predicting the reverse water-hammer problem in the draft-tube and analyzing the mechanism of this process.

2. Numerical methods and validation

2.1. Numerical methods

2.1.1. 1D-3D method for hydraulic transients

In this study, the 1D-3D coupling transient simulation approach is adopted for its high accuracy and efficiency. The basic idea of 1D-3D coupling is to divide the computational domain into two parts according to flow patterns. The parts of gradually varied flow, such as pipes and tunnels, are simplified to 1D elements and calculated by the Method of Characteristics (MOC). With regard to the parts of rapidly varied flow, such as hydro-turbines and surge tanks, the 3D CFD are implemented. At present, there are several coupling schemes, among which the partly overlapped coupling (POC) scheme[9] is physically reliable and chosen here. Detailed procedures and applications about this scheme can be

found in reference [9], [10] and [11].

2.1.2. Cavitation models

There are many cavitation models available in CFD simulation for engineering purpose, such as Singhal et al. model[12], Zwart-Gerber-Belamri model[13] and Schnerr & Sauer model[14]. The first one is also known as Full Cavitation Model, which has been embedded in many commercial codes. But this model is numerically unstable and leads to divergence more easily. The latter two are implemented following an entirely different numerical procedure from the first model and perform better in convergence, which are highly recommended by Ansys Fluent[15]. Therefore, considering the numerical challenges in simulating transient cavitating flow in the pump-turbine, we chose the Schnerr & Sauer model.

2.2. Validation

2.2.1. Computational conditions

The typical water-hammer in a tank-pipe-valve system induced by instantaneous valve closure is simulated to validate the 1D-3D coupling approach and Schnerr & Sauer cavitation model. The test case[16] includes an upper tank, a lower tank and a sloping pipe with a valve at the higher side, which makes it form vapor cavities more easily at the valve during the pressure dropping processes. The computational domain is simplified as shown in figure 1, in which the 3D zone is simulated by Ansys Fluent 12.0 and the 1D zone is solved by the User Defined Function (UDF) in Fluent. Initially, the flow velocity in the pipe is 0.3m/s and the valve is closed within 0.009s. During the pressure variation process, the water level in the upper tank is fixed as 22.0 m. Assuming that there just generates local large vapor cavities and the gas content in the water is invariable, so the acoustic speed of the water can be treated as constant, as 1319 m/s.

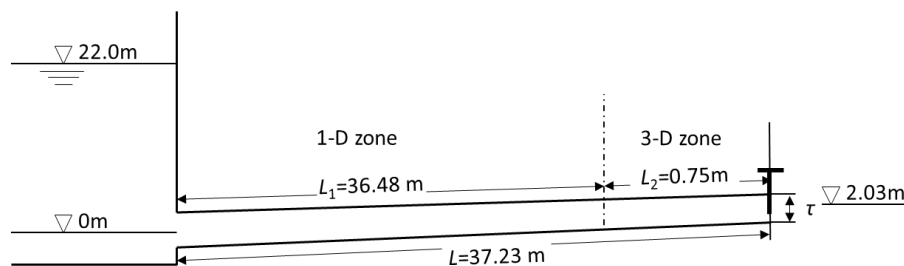


Figure 1. Calculation diagram of the model test apparatus built by Bergant et al.[16]

2.2.2. Comparison of numerical and measured data

By virtue of the Schnerr & Sauer cavitation model and the 1D-3D coupling approach, the 3D method is able to capture the reverse water-hammer accurately. In figure 2, comparisons of the simulated valve head among the 3D CFD, the 1D MOC (by the Discrete Vapor Cavity Model[17]), and the measured data from the model test[16] are conducted. All of the curves experience short-duration pressure peaks and show good agreements, which indicates that the 3D CFD is reliable qualitatively. Additionally, comparisons of the key parameters among the three methods (Table 1) demonstrate that the results of the 3D CFD are closer to the measured ones than that of the 1D method. Moreover, the detailed flow

patterns given by the 3D CFD (in figure 3) show the dynamic developments of the cavity clearly: at point B, the pressure at the valve reduces to the vapor pressure and then produces a cavity; after that the pressure maintains the same but the cavity expands with the effect of the negative flow (from the valve to the tank); at point C, the flow has converted to the positive direction (from the tank to the valve) and the cavity begins to shrink. Finally, the pressure-rising wave propagates to the valve, making the cavity collapse. The water flow strikes the valve directly and causes a short-duration pressure peaks (point E).

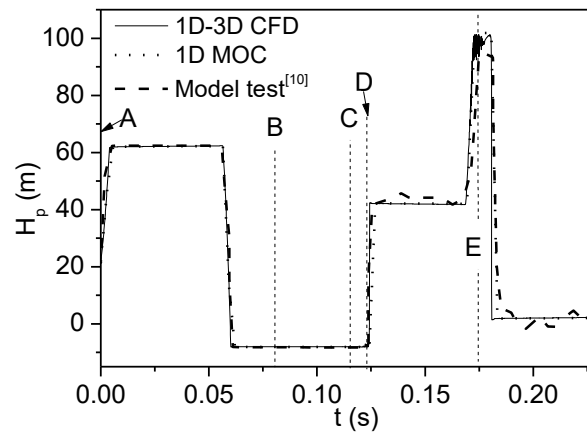


Figure 2. Comparisons of the obtained valve heads among 1D-3D CFD, 1D MOC and model test

Table 1 Comparisons of the obtained key parameters among 1D-3D CFD, 1D MOC and model test

	1D-3D CFD	1D MOC	Model test[16]
Time of the cavity formation (s)	0.0603	0.0621	0.0612
Time of the cavity collapse (s)	0.1244	0.1270	0.1248
Time of the maximal pressure appearing (s)	0.1800	0.1750	0.1792
Value of the maximal pressure (m)	101.3	101.8	95.6

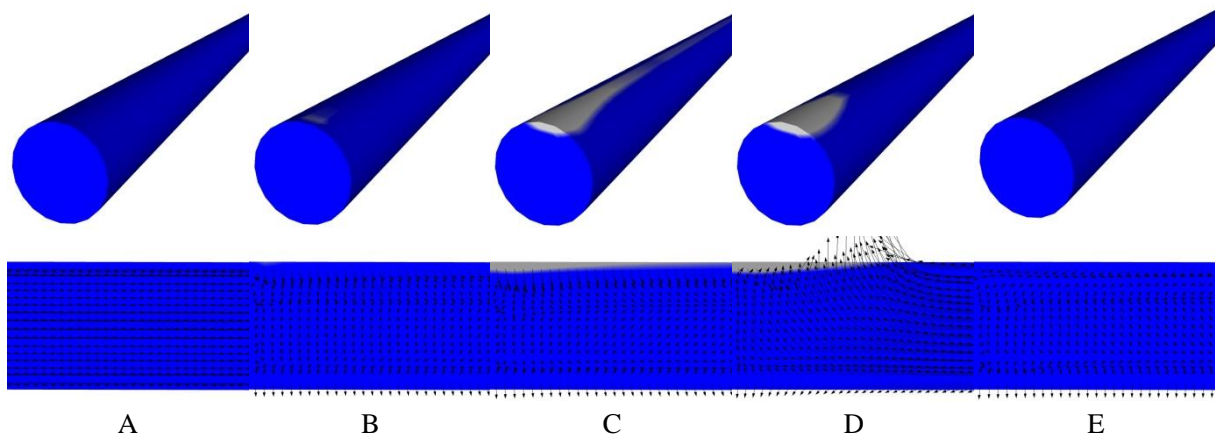


Figure 3 Flow patterns at the valve (Blue zones means water and gray zones means cavity)

Therefore, the 3D method adopted here is feasible in achieving not only the accurate transient pressure during reverse water-hammer, but also the detailed flow patterns, which contributes significantly to the

analysis of more complicated transient phenomenon in hydraulic machines.

3. Cavity evolution and reverse water-hammer in the draft-tube of a model pump-turbine

3.1. Model pumped-storage system

The model pumped storage system considered here includes one penstock, one pump-turbine and one tailrace tunnel with a surge tank, as shown in figure 4. In this case, the 1D-3D coupling approach is also adopted: the pump-turbine is simulated by the 3D CFD while the water conveyance system is modeled by 1D MOC. In this approach, discharge obtained from the 3D CFD is set as the boundary condition of the 1D zone and pressure calculated by the 1D MOC is set as the boundary condition of the 3D zone. To obtain a uniformly distributed flow on the coupling boundaries, parts of the flow passages in front of the spiral casing and behind the draft tube are also treated by the 3D model.

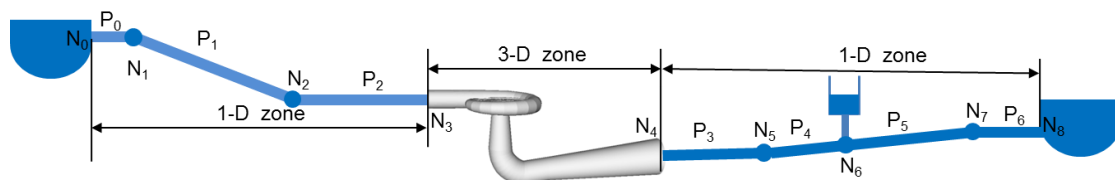


Figure 4. Schematic diagram of the whole computational domain

3.2. Computational conditions

During the unit acceleration and deceleration period, the moving mesh model is adopted in the rotational zone and the angular speed is defined according to the hydraulic torque of the runner. For 3D pump-turbine, the mesh sensitivity is investigated first and a mesh scheme with about 4,300,000 cells is finally chosen. For turbulence simulation, the ν^2 - f model is adopted for its advantage in simulating flow dominated by separations[18]. During the simulation, the time step is fixed to 4×10^{-4} s and the convergence criteria is set as 1×10^{-5} according to our previous studies[19].

Initially, the pump-turbine operates in turbine mode with the rotational speed of 1000 rpm and the discharge of $0.049 \text{ m}^3/\text{s}$. The setting elevation of the pump-turbine is increased by 8.93 m, relative to the downstream water level, in order to induce cavity more easily in the draft-tube.

In this paper, we just focus on the reverse water-hammer of the pump-turbine during runaway, which shows cavitation and pressure variations in the draft-tube inlet, but the detailed validation of the above model setup and the in-depth analysis of the transients can be found in reference [19].

3.3. Results and discussion

3.3.1. Flow characteristics

Runaway occurs when the electromagnetic load of the pump-turbine dropping to zero immediately and the hydraulic energy is only used to accelerate the runner. Then, the operating point deviates from the rated point and runs towards the runaway point. This generates variations in pressure and induces cavitation at the draft-tube inlet. The dynamic operating trajectories defined by unit parameters are depicted in figure 5, in which the gray rectangular area represents where cavitation happens at the draft-tube inlet.

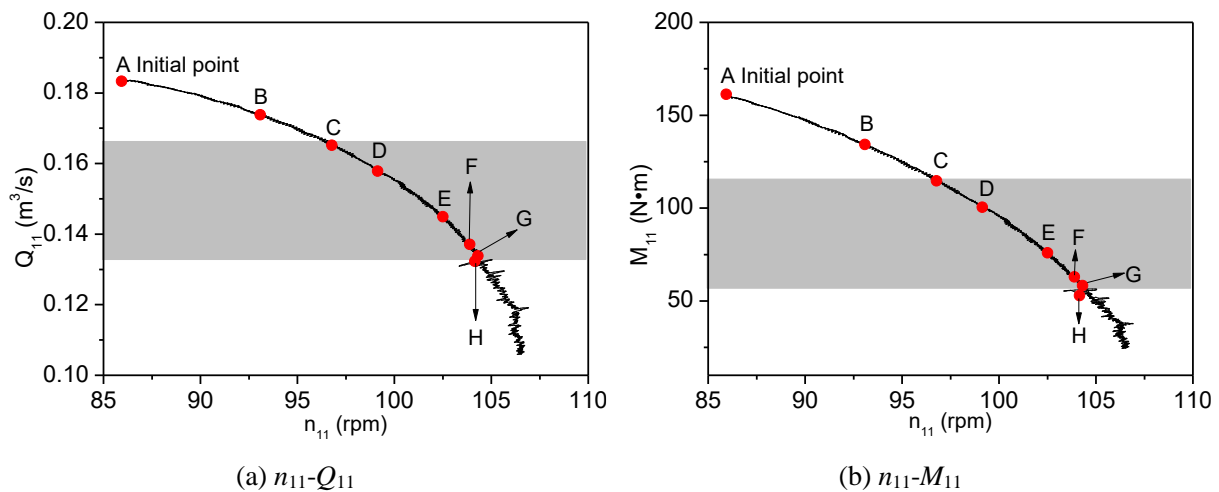


Figure 5. Dynamic trajectories of the pump-turbine during runaway (The gray rectangular area represents where cavitation happens in the draft-tube)

Reverse water-hammer includes variations in pressure, flow direction and cavity, which needs comprehensive analysis. The pressure monitored in the center of the runner cone as well as the flow patterns in the vertical section of the draft-tube are shown in figure 6, in which the color contours represent the pressure and the color of the velocity vectors means the flow directions: white means the positive flow (from upstream to downstream) and black means the negative flow (from downstream to upstream). Figure 7 presents the variations of the volume and the shape (the bottom radius r and the axial length L) of the cavity, in order to analyze of the cavity quantitatively.

Initially (point A), the pressure in the draft-tube distributes uniformly and the flow direction is positive. With the runaway going on, the distribution of the pressure nears the inlet of the draft-tube changes: the pressure decreases in the central region and increases along the radial direction from inside to outside. This phenomenon is caused by both the flow characteristics in part load operating conditions and the pressure-reducing wave generated by the acceleration of the runner. As a result, the pressure near the draft-tube inlet is significantly lower than other parts and there forms a low pressure region under the runner cone, as shown at point B. The pressure in low pressure region reduces to vaporization value and causes a conical cavity attaching to the runner cone at the end, at point C. With the increase of the scope of low pressure, the volume, the axis length and radius of the cavity gets larger. At point D, the axis length of the cavity reaches its maximal value and then starts to decline, but the radius and the volume of the cavity keep increasing. At point E, the cavity expands to its full volume and wraps around the bottom of the runner cone. Meanwhile, negative flow in the draft-tube occurs, indicating that the arrival of the pressure-increasing wave reflected by the lower tank. With the increase of the intensity of the reverse flow, the volume of the cavity decreases but the radius keeps increasing. After about 0.80 s (point F), the volume and axial length of the cavity drops sharply but the radius still keeps increasing. After 0.04 s (point G), the cavity shrink to a thin vapor film attaching to the bottom of the runner cone and the radius reaches its maximal size. Then the radius decrease sharply and the cavity collapse in 0.01 s. At the same time, the reverse water flow strikes the runner cone, causing sudden pressure augment (point H).

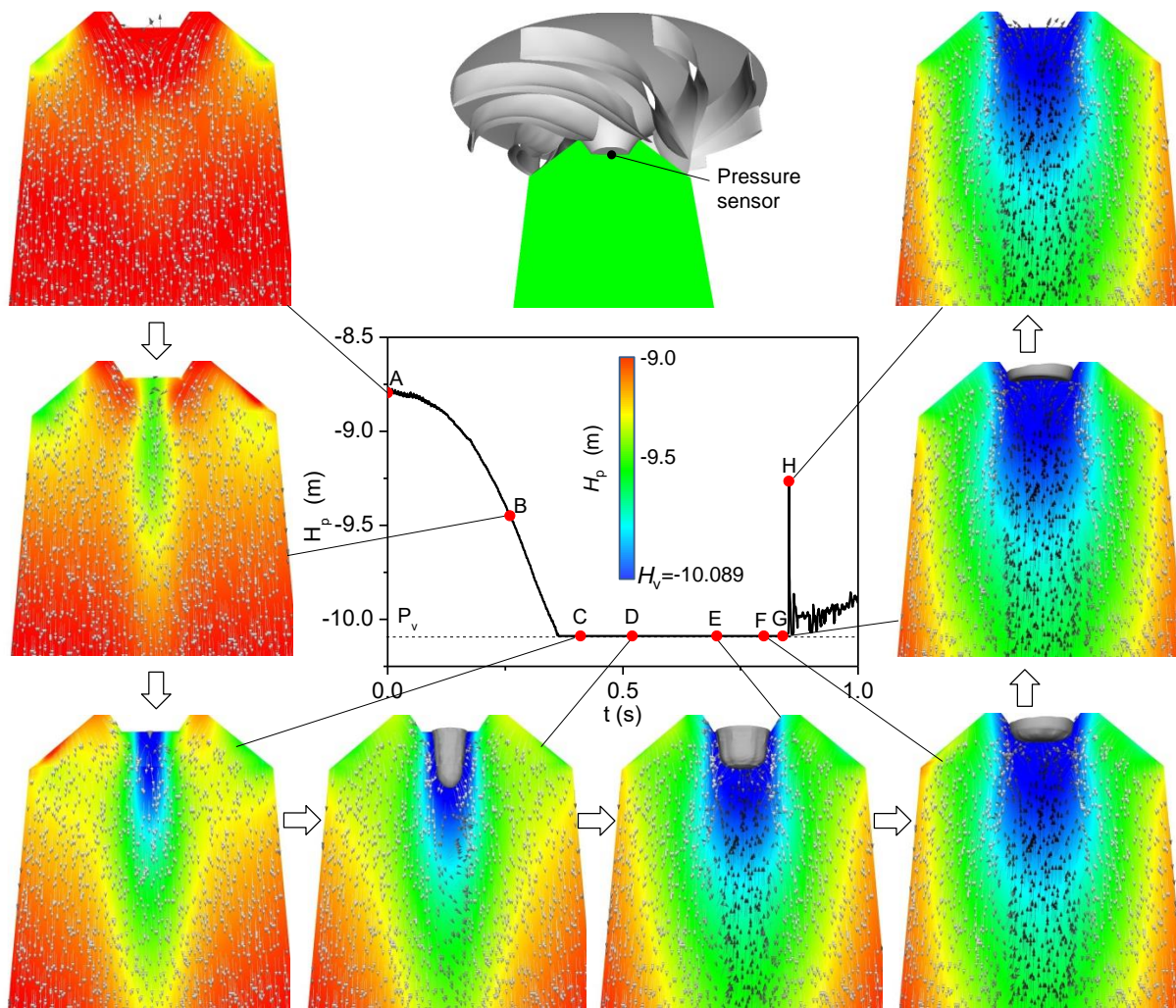


Figure 6. Variations of the operating parameters during the runaway process

The above description can be divided into two stages. The first one is the generation, development and collapse of the local vapor cavity, and the other one is the turning of the flow (from positive to negative) under the runner cone. The negative flow strikes the runner cone and sudden pressure rising happens, causing reverse water-hammer.

3.3.2. Pressure evolution

The reverse water-hammer in the draft-tube of the pump-turbine is characterized by the local vapor cavity, which relates to the temporal and spatial characteristics of the pressure at the inlet of the draft-tube of the pump-turbine, as shown in figure 8. At the rated point (point A), the pressure in the center is the largest and decreases gradually along the radius from the center. With the operating point deviating from the rated point, the flow patterns in the draft-tube begin to change as well as the pressure distributions. On one hand, the turbine discharge is decreased by the acceleration of the runner, which produces pressure-reducing wave in the draft-tube. As a result, the pressure at the inlet of the draft-tube is lowered with time overall. On the other hand, the pressure varies spatially. Specifically, the outer pressure increases while the inner pressure decreases along the radius, causing the pressure at the

center reaches the vapor value firstly. Consequently, the cavitation is restricted in the central region of the draft-tube inlet and not able to shut off the water flow overall.

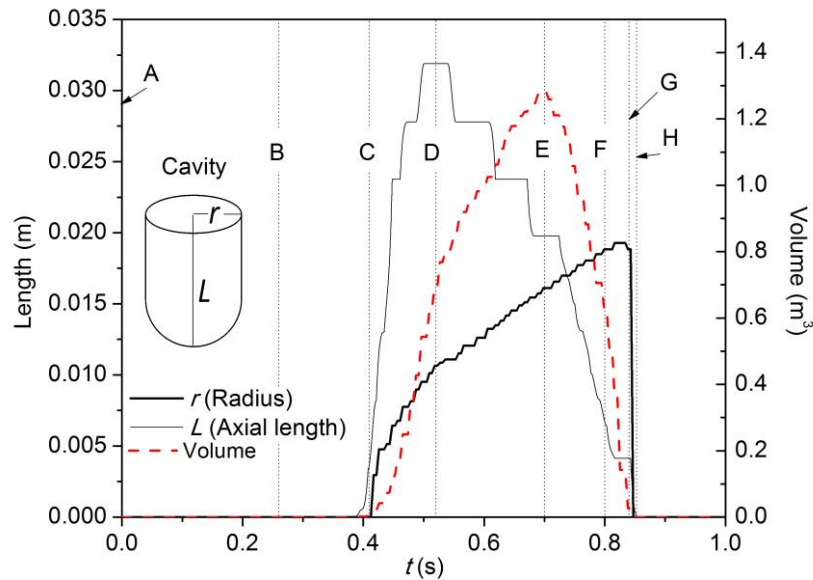


Figure 7. Temporal variations of the radius and axial length of the cavity

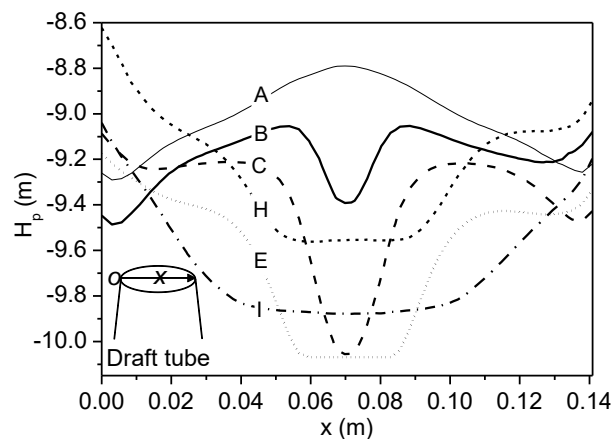


Figure 8. Temporal and spatial variations of the pressure at the inlet of the draft-tube

4. Conclusions

The reverse water-hammer induced by the collapse of draft-tube cavity in a model pump-turbine during the runaway process is simulated successfully by 3D CFD. The results show the dynamic process of a vapor cavity, from generation, expansion, shrink to collapse. After the cavity collapsed, a sudden increase of pressure can be evidently observed. These phenomena are in accordance with the basic elements of the reverse water-hammer, which confirms the validity of the 3D method qualitatively. Besides, from the present simulation, we found that the reverse water-hammer in pump-turbine during runaway is featured by a locally expanding and collapsing vapor cavity that is around the runner cone, which is different from the conventional recognition of violent water-column separation.

This work reveals the possibility for simulating the reverse water-hammer phenomenon in a pump-turbine by 3D CFD. Future work should be focused on the quantitative validation of the cavitating flow patterns simulated in this study and the optimization of the mathematic models adopted here. Additionally, more hydraulic characteristics of pump-turbine during reverse water-hammer (such as the pressure pulsation and axial thrust) and more dangerous operating conditions, especially the load rejection process, still need to be investigated.

Acknowledgements

The authors would like to thank the National Natural Science Foundation of China (NSFC, No. 51579187 and 51039005) for their financial support.

Nomenclature

D_1 (m)	runner diameter	H (m)	head
H_d (m)	pressure head	H_v (m)	vapor pressure head
M (N·m)	hydraulic torque	M_{11} (N·m)	unit torque ($=M/(HD_1^3)$)
n (rpm)	angular rotational speed	n_{11} (rpm)	unit speed ($=n D_1/\sqrt{H}$)
Q (m ³ /s)	discharge	Q_{11} (m ³ /s)	unit discharge ($=Q/(\sqrt{H} D_1^2)$)

Reference

- [1] Bergant A, Simpson A R and Tijsseling A S 2006 *J. Fluid. Struct.* **22** 135
- [2] Nicolet C, Alligne S, Bergant A and Avellan F 2012 *Houille Blanche-Revue Internationale De L Eau* p 44
- [3] Nicolet C, Kaelbel T, Alligne S, Ruchonnet N, Allenbach P, Bergant A and Avellan F 2011 *4th IAHR Int. Meeting of the WorkGroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems* (Belgrade, Serbia)
- [4] Nicolet C, Alligne S, Bergant A and Avellan F 2012 *26th IAHR Symp. on Hydraulic Machinery and Systems* (Beijing, China)
- [5] Pejovic S, Zhang Q F, Karney B and Gajic A 2011 *4th Int. Meeting on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems* (Belgrade, Serbia)
- [6] Liu S, Zhang L, Nishi M and Wu Yu 2009 *J. Fluid. Eng-t. Asme.* 2009 **131** 513021
- [7] Jošt D and Lipej A 2011 *Strojniški vestnik-J. Mech. Eng.* **57** 445
- [8] Kumar P and Saini R P 2010 *Renew. Sust. Energ. Rev.* **14** 374
- [9] Zhang X and Cheng Y 2012 *J. Hydrodynamics* **24** 595
- [10] Zhang X, Cheng Y, Yang J, Xia L and Lai X 2014 *J. Hydrodynamics* **26** 715
- [11] Zhang X, Cheng Y, Xia L and Yang J 2014 *27th IAHR Symp. on Hydraulic Machinery and Systems* (Montreal, Canada)
- [12] Singhal A K, Athavale M M, Li H Y and Jian Y 2002 *J. Fluid. Eng-t. Asme.* **124** 617
- [13] Zwart P J, Gerber A G and Belamri T 2004 *5th Int. Conf. on Multiphase Flow* (Yokohama, Japan)
- [14] Schnerr G H and Sauer J 2001 *4th Int. Conf. on Multiphase Flow* (New Orleans, USA)
- [15] Ansys, Inc 2009 Ansys Fluent 12.0 Theory Guide (Canonsburg, PA, USA)
- [16] Bergant A and Simpson A R 1999 *J. Hydraul. Eng-asce.* **125** 835
- [17] Wylie E B, Streeter V L and Suo L 1993 *Fluid transients in systems* (Englewood Cliffs, NJ: Prentice Hall)

[18] Durbin P A 1995 *AIAA J.* **33** 659

[19] Zhang X, Cheng Y, Xia L, Yang J and Qian Z 2016 *J. Fluid. Eng-t. Asme.* **138** 091102