Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2017,29(6):972-978
DOI: 10.1016/S1001-6058(16)60811-9
www.sciencedirect.com/science/journal/l0016058

Simulation of cavitation induced by water hammer *

Jie Geng (耿介), Xiu-le Yuan (苑修乐), Dong Li (李冬), Guang-sheng Du (杜广生)
School of Energy and Power Engineering, Shandong University, Jinan 250061, China,
E-mail: gj_8944@163.com

(Received May 23, 2016, Revised September 7, 2016)

Abstract: A simulation is carried out for the pressure fluctuation driven by the water hammer, based on a joint use of the one-dimensional method of characteristics (MOC) and the three-dimensional finite volume method (FVM). The three-dimensional visualization of the cavitation induced by the water hammer is implemented, and the temporal and spatial analyses of extreme regions are made. A practical case of the water hammer, with the minimum boundary pressure higher than the saturated vapor pressure condition, is simulated. The simulation prediction that the cavitation would occur in the front of the gasket could serve some guideline for the optimization of industrial designs.

Key words: Water hammer, cavitation, method of characteristics (MOC), finite volume method (FVM)

Introduction

The cavitation might happen when the local pressure is lower than the working fluid's saturated vapor pressure, which is a discontinuous process. The cavitation widely exists in nature, such as, behind blades of the rotor propeller, under the waterfall and on the surface of rocks in high-speed rivers[1,2].

In the hydraulic system, the cavitation generally consists of four processes, the gas nuclear generation, the expansion, the compression, and the collapse process. The noise and the vibration, caused by the pressure pulsation with bubble collapsing, are harmful in most cases. The cavitation will seriously affect the reliability and durability[1]. Therefore, the prevention of the cavitation is an important issue in the industrial designs. The water hammer can create a pressure transient in the pipeline, which may lead to the cavitation[3]. The phenomenon is often induced by an instantaneous closure of the valve. Nowadays, with the increasing demands for the precise flow control, a high speed valve is very common. Consequently, the water hammer prevention becomes more and more important. Since the nineteenth century when Russian scientist Joukowski (1898) firstly introduced the theory of water hammer, the water hammer has been a research focus for more than a hundred years. The main numerical methods include: the MOC, the wave front method, the finite difference (FD) method, and the finite volume method (FVM). The MOC is the most popular technique for solving partial differential equations for the water hammer, with a high computational efficiency, and a programming simplicity. Acrivos[4] developed the method of characteristic (MOC) for heat and mass transfer problems. Tijsseling and Bergant[5] removed the need of grid development by introducing the meshless method. An Implicit Method of Characteristics is proposed by Afshar and Rohani[6] and any arbitrary combination of devices can be allowed with implicit method. Zhao and Ghidaoui[7] compared the application of FVM schemes and MOC schemes with space line interpolation and found that MOC produces the same results with first-order FV Godunov-scheme. Meniconi et al.[8] studied the interaction between valve action and an in-line devices using MOC. The predictions with the MOC are relatively reliable, widely used in energy plants, environment industry, agriculture automation, chemical industry, urban water supply and other fields.

The water hammer is a complex process, and the simulation is mostly limited to one-dimensional cases[9].The industrial design is restricted by an unreal dimension. With the progress of computer science and

* Project supported by the Natural Science Foundation of Shandong Province (Grant No. ZR2014ZZM015).
Biography: Jie Geng (1989-), Male, Ph. D. Candidate
Corresponding author: Guang-sheng Du,
E-mail: du@sdu.edu.cn

hardware, the computational cost is greatly reduced. Many approaches of three-dimensional cavitation simulation with the FVM were proposed in recent years. A general cavitation prediction method was developed by Singhal et al.[10] and introduced to improve the performance of turbo-machinery. Bakir et al.[11] investigated the cavitating behavior of an Inducer. Shang et al.[12] studied the cavitation around a high speed submarine. To the best of the authors' knowledge, the related references mainly focused on the cases of pumps and high speed airfoils[13-18], the cavitation induced by the water hammer has not yet been studied by using three-dimensional numerical methods.

In the following, joint simulation by one-dimensional water hammer software "Hammer" and open source FVM software "OpenFOAM" is introduced. Firstly, pressure changes on the boundary will be calculated by MOC with Hammer. Then, the three-dimensional prediction of cavitation will be achieved and three-dimensional visualization will be realized with the help of FVM. By analysis of the causes of cavitation from a three-dimensional perspective, it can provide new ideas for the industrial design of the fluid equipment.

1. Model description

1.1 *Equations of water hammer*

The propagation velocity of the wave that would induce the water hammer[4]

$$c = \sqrt{\frac{E_v}{\rho\left[1+\left(\frac{D}{T}\right)\left(\frac{E_v}{E}\right)\right]}} \tag{1}$$

where E_v is the bulk modulus of liquid, E is the elastic modulus of tube's wall, D is the pipe diameter, and T is the wall thickness.

The basic differential equations of the water hammer[19,20]:

$$\frac{\partial H}{\partial t} + v\frac{\partial H}{\partial s} + \frac{c^2}{g}\frac{\partial v}{\partial s} + v\sin\alpha = 0 \tag{2}$$

$$\frac{\partial H}{\partial s} + \frac{1}{g}\left(\frac{\partial v}{\partial t} + v\frac{\partial v}{\partial s}\right) + \frac{\lambda|v|v}{8gR} = 0 \tag{3}$$

where H is the static pressure head, α is the inclined angle of the pipeline, λ is the resistance coefficient and s is the axial length.

1.2 *Equations of turbulence and cavitation*

The k-ω-SST model[21] is used to model the turbulence, which enjoys a robust convergence rate and a relatively accurate inverse pressure gradient and separated vortex. The cavitation model is based on the volume of fluid model (VOF), the interPhaseChangeFoam in OpenFOAM.

The k and ω equations in the k-ω-SST model:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = P - \beta^*\rho\omega k + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_k\mu_t)\frac{\partial k}{x_j}\right] \tag{4}$$

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho u_j\omega)}{\partial x_j} = \frac{\gamma}{\nu_t}P - \beta\rho\omega^2 +$$

$$\frac{\partial}{\partial x_j}\left[(\mu + \sigma_\omega\mu_t)\frac{\partial\omega}{x_j}\right] + 2(1-F_1)\frac{\rho\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j} \tag{5}$$

where

$$P = T_{ij}\frac{u_i}{x_j}\ ,\ T_{ij} = \mu_t\left(2S_{ij} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right) - \frac{2}{3}\rho k\delta_{ij},$$

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

The phase transition is described by the Merkle mass transformation equation:

$$\dot{m}^- = \frac{C_v\rho_v}{\frac{1}{2}\rho_1 U_\infty^2 t_\infty}\alpha\min(0, p - p_{\text{sat}}) \tag{6}$$

$$\dot{m}^+ = \frac{C_c}{\frac{1}{2}U_\infty^2 t_\infty}(1-\alpha)\max(0, p - p_{\text{sat}}) \tag{7}$$

where $\dot{m}^-$ is the transition rate from the liquid phase to the gas phase, and $\dot{m}^+$ is the transition rate from the gas phase to the liquid phase, C_c, C_v, t_∞, U_∞ are constants depending on the mean velocity, and p_{sat} is the saturated vapor pressure.

1.3 *Boundary condition and grid settings*

The physical model, initially designed by Nicolaus Bernoulli, is a flow measurement system operated in the Fluid Dynamics Laboratory of Shandong University[13,22,23], as shown in Fig.1.

High speed valves are used to ensure the accuracy of the flow control (the valve closing process involves nonlinear changes of the flow rate. Improving