V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

CFD NUMERICAL SIMULATION OF WATER HAMMER IN PIPELINE BASED ON THE NAVIER-STOKES EQUATION

LI Jinping*, WU Peng†, and YANG Jiandong†

*Water Resources and Hydropower Engineering Science State Key Laboratory, Wuhan University, Wuhan, Hubei, China
e-mail: lukeping@whu.edu.cn

†Water Resources and Hydropower Engineering Science State Key Laboratory, Wuhan University, Wuhan, Hubei, China
e-mail: wupengaiyou@163.com, jdyang@whu.edu.cn

Key words: Water hammer, Weak-compressible flow, Pipeline, Numerical algorithm

Abstract: *Directly starting from the Navier-Stokes Equations, taking into account the water elasticity and pipeline flexibility, the SIMPLE series numerical algorithm for weakly compressible water is derived from the constitutive equation of water and the principle of SIMPLE series algorithms. Then the water hammer is calculated by 3D flow simulation in water pipeline system. It is a system which has a large reservoir with constant pressure at one end and a quick closing valve at the other end, and there are measurements of Bergant & Simpson (1991). Then the simulation method can be verified by comparing the numerical results of CFD with the model test result. On base of this result, the water hammer in pressure pipeline, and its influence factor, can be analyzed. The sensitivity analysis of the unsteady friction, the valve closing law and the velocity of hydro acoustic wave can be launched and the results can provides the base for the 3D unsteady flow simulation for the water conduits of the hydropower system.*

1 INTRODUCTION

In a hydropower station, the hydro-turbine frequently adjusts the discharge according to the electricity load. In some cases the unit will shutdown in emergency, and the "water hammer" phenomenon will inevitable happen in the pressure pipeline (penstock). This phenomenon may occur in all of pressure pipeline system, often bringing about strong vibration and damage on the pipeline. Therefore, the calculation of water hammer plays an important role in the design and operation of a hydropower station (in a broad scope, including all pressurized piping systems).

The water hammer is characterized by that, when a little change of water velocity (discharge) occurs, there will produce significant rise and drop of pressure in the pipeline. The water hammer propagates as elastic wave along the pipeline. At present, this transient flow (water hammer) is simulated by the method of one-dimensional method of characteristics. And the weak compressibility and wall elasticity must be considered in the numerical simulation process. However, during the research of transient process of the hydraulic machinery and systems, the traditional one-dimensional method cannot attain adequate accuracy. The three-dimensional features must be considered in order to accurately simulate the system characteristics.

Therefore, based on fundamental Navier-Stokes equation, considering the flexibility of water and pipeline, this paper is to explore the CFD-based numerical method for water hammer.

2 MATHEMATICAL MODEL

For the unsteady flow calculation of compressible fluid, it is usually to establish the corresponding mathematical models and numerical methods from the basic Navier-Stokes equations. In this respect, there are a lot of achievements in aerodynamics, substantially promoting the development of other areas. In the simulation of water hammer, the temperature generally is not taken into account, and the water viscosity, compressibility and some physical properties is assumed to be independent of temperature, so the energy equation can be neglected.

The mathematical model used in this paper is as follows:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + div\left(\rho \vec{u}\right) = 0 \tag{1}$$

Momentum equation:

$$\left.\begin{aligned} \frac{\partial(\rho u)}{\partial t} + div\left(\rho u \vec{u}\right) &= -\frac{\partial p}{\partial x} + div\left(\mu \; grad \; u\right) + S_u \\ \frac{\partial(\rho v)}{\partial t} + div\left(\rho v \vec{u}\right) &= -\frac{\partial p}{\partial x} + div\left(\mu \; grad \; v\right) + S_v \\ \frac{\partial(\rho w)}{\partial t} + div\left(\rho w \vec{u}\right) &= -\frac{\partial p}{\partial x} + div\left(\mu \; grad \; w\right) + S_w \end{aligned}\right\} \tag{2}$$

Constitutive equation：

$$\frac{d\rho}{\rho} = \frac{dp}{K} \tag{3}$$

$$a^2 = \frac{\partial p}{\partial \rho} \tag{4}$$

According to the equation (3), the compressibility of water can be considered. Equation (4) is the formula for the sound velocity of small perturbation in the unbounded water. In fact, the sound velocity is dependent on the water elasticity and the pipeline flexibility. During the simulation, the body volume modulus of water will be adjusted to fit the sound velocity in pipeline.

3 NUMERICAL METHODE

The algorithm of SIMPLE series has been widely applied in the incompressible turbulence flow. This method can also be applied to the compressible flow with any Mach number by modifying the pressure-correction equation and introducing the constitute equation to the continuity equation[1 ~ 3]. It is beneficial to use this numerical method to simulation the water hammer here. There is available existing code, and it is just need to introduce pressure-density relation of water to the continuity equation.

In the algorithm of SIMPLE series for compressible flow, the relationship of velocity- pressure- density is derived from the continuity equation. The relationship of pressure – density is derived from the constitutive equation (or status equation). Then, the pressure correction equation can be obtained by coupling these two equations[4 ~ 7].

The calculation steps of the SIMPLE algorithm for compressible are as follows:

(1) Given the initial pressure field and density field, the new velocity field can be calculated by solving the momentum equation.

(2) The correction value can be calculated by solving pressure correction equation, then the new pressure field and velocity field can be obtained.

(3) The density field can be calculated by the constitutive equation of water from the new density field.

(4) All of the other variables and coefficients of the discrete equations can be obtained by solving the turbulence model equations.

(5) If the convergence condition achieves, the calculation steps stop. Otherwise, return to step (1), star the next iteration.

4 MODEL VERIFICATION

4.1 Calculation Condition

Figure 1 is the test equipment that Bergant and Simpson[8] (1991) had done the water hammer test on it. The test device consists of the upper water tank, copper pipe, fast closing valve and downstream water tank. The pipe is connected to the two tanks with a little tilt, length 37.2m, diameter 22mm, and wall thickness 1.63mm. The fast closing valve located at the end of the pipe.

This model test is simulated by CFD, and the results are compared with the experimental data to verify the mathematical model and numerical method.

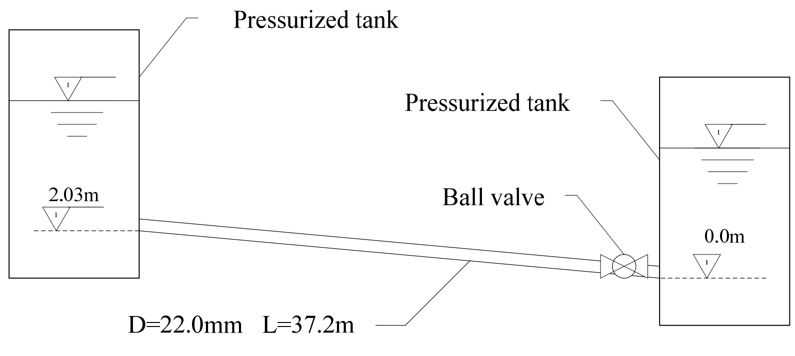

Figure 1: Bergant water hammer test device

The basic parameters of the water hammer experiment are as follows:

Initial velocity in the pipe is 0.3m/s, the pressure at the upper tank is 32.0m (water head), the closing time of the copper ball valve is 0.009s, the velocity of the water hammer wave is 1319m/s, the Reynolds number is 5600, and the water temperature is 15.5 degree during the experiment.

4.2 Numerical Simulation

The experimental pipe is simulated by CFD. The upstream is the pressure inlet boundary, given the pressure of the upper tank. The downstream is the velocity output boundary, given initial velocity as 0.3m/s. After the steady iteration, the downstream boundary is swift to wall boundary so that the water hammer phenomena is achieved. The pipe wall is set to solid wall boundary with non- slip condition. The wall function is used to simulate the boundary layer.

The RNG $k-\varepsilon$ turbulent model is selected, and the discrete scheme is finite volume method (Finite Volume Method). In order to understand the influence of the discrete scheme on the numerical simulation, three different combinations have been chosen to do some comparison. Table 1 shows the combination of the discrete format of convection item, discrete format transient item, the interpolation scheme of interface pressure in the SIMPLE series algorithm. During the transient flow calculation, the time step is 0.01s.

Case	Transient item	Convective item	Pressure interpolation	Diffusion item
1	First order implicit	Third order MUSCL	Momentum Interpolation	Central difference
2	First order implicit	Third order MUSCL	Second order	Central difference
3	Second order implicit	First order upwind scheme	Momentum Interpolation	Central difference

Table 1 Combination of different numerical formats

4.3 Results and Analysis

(1) The pressure fluctuation at the vale and the velocity fluctuation at the inlet

Figure 2 shows the pressure fluctuation at the valve including the experimental data. Figure 3 shows the fluctuation of average cross-section velocity at the inlet. All of the numerical results, not only the pressure but also the velocity, have a high coincidence with each other. It is concluded that, the discretization scheme of all items has little influence on the results of water hammer simulation. So a further study must be done to understand the difference with the experiment.

From the pressure fluctuation at the valve, we can know that, the maximum and minimum water hammer pressure of numerical and experimental data is very consistent at the first propagation stage of water hammer wave. But with the time increasing (the water hammer is propagating), the distortion between the two data is getting bigger, and the wave front of the water hammer is smoothing.

We think that the main cause of the distortion is the numerical dissipation (false diffusion), instead of the discretization scheme. Otherwise, the time step in the unsteady numerical simulation is another important reason. In this problem, the propagation period of the water-hammer wave is 0.0564s, but the time step is 0.001s. Therefore, there are only 28 calculations in one half-wave process, and it is not enough to predict the steep wave front. So, the time step should be less than $10^{-2} \sim 10^{-3}$ of the period (further study is in section 5).

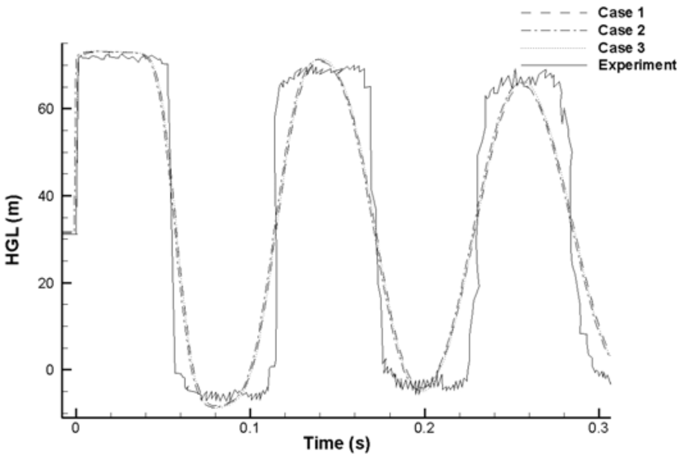

Figure 2: pressure fluctuations at the valve(outlet)

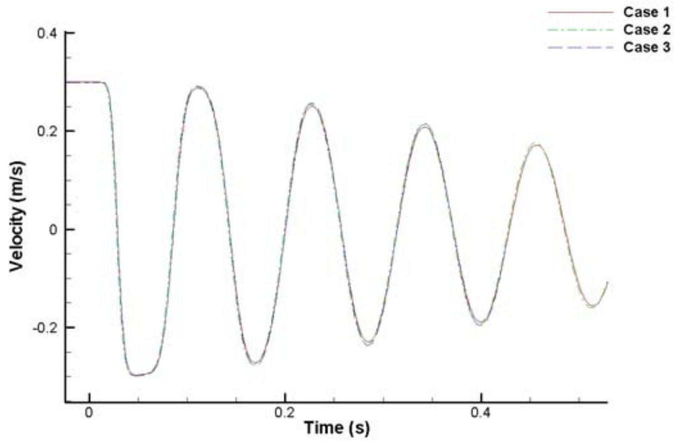

Figure 3: water velocity fluctuations at the inlet

(2) The distribution of velocity at the middle cross section of the pipe

In order to understand the unsteady character of veolocity distribution during the water hammer wave propagation, one vertical diameter of the cross section at the middle of the pipe has chozen to illustrate the numerical results. Figure 4 shows the velocity distribution from 0.01s to 0.06s. And Figure 5 shows the velocity distribution from 0.07s to 0.12s. In the figures, when the velocity directed to downstream, the value is positive.

According to theoretical analysis, during one period, the water hammer will propagate through the middle cross-section four times. The first time is 0.014s, second time is 0.042s, third time is 0.07s, and fourth time is 0.098s, and so on.

When t=0.01s, the first positive wave has not reach the middle section, so the distribution is the same to the initial status.

When t=0.02s, 0.03s, 0.04s, the first positive wave has past the middle section, but the second negative has not arrived. In one dimensional theory (no consider the friction), the averaged velocity in cross-section should be 0m/s during this time. In fact, we can see that there is a delay, and the averaged velocity has a process from a little positive to a little negative.

When t=0.05s, 0.06s, 0.07s, the second negative has past the middle section, but the third negative wave has not past. In one dimensional theory, the averaged velocity in cross section should be negative, and the absolute value equals the initial one. There also is a delayed variation during this time.

When t=0.08s, 0.09s, the third negative wave has past the middle section, but the fourth positive wave has not arrived. So the averaged velocity in cross section should be 0m/s again. But there still is a delay, and the averaged velocity has a process from a little negative to a little positive.

When t=0.1s, 0.11s, 0.12s, the fourth positive wave has past the middle section. So, the averaged velocity in cross section should be the initial value again. The numerical results of CFD shows there still a delay. The variation is not happen instantaneously when the wave reaches this section.

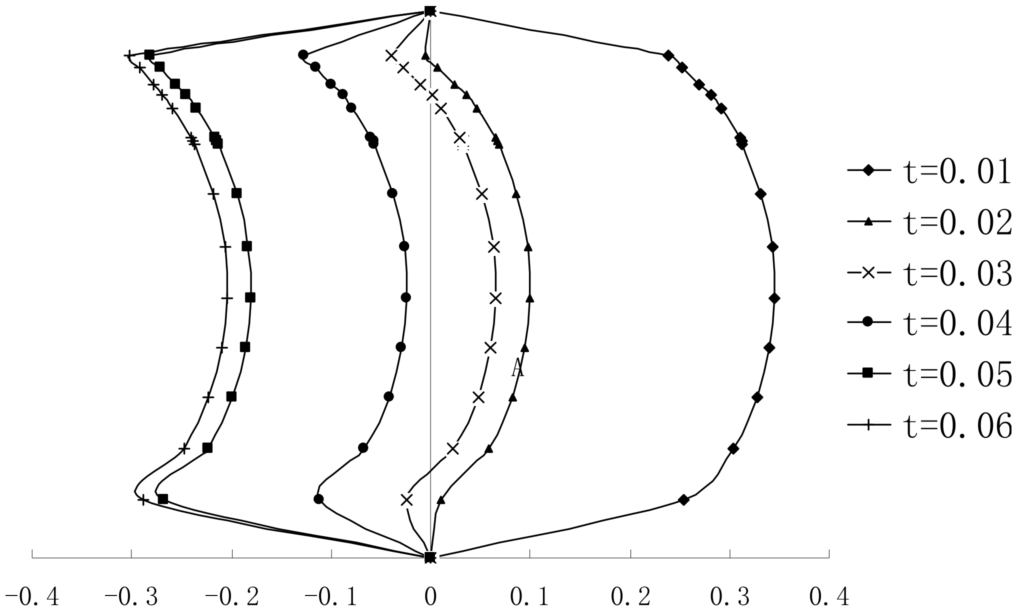

Figure 5: velocity ditribution in cross-section from 0.07s to 0.12s

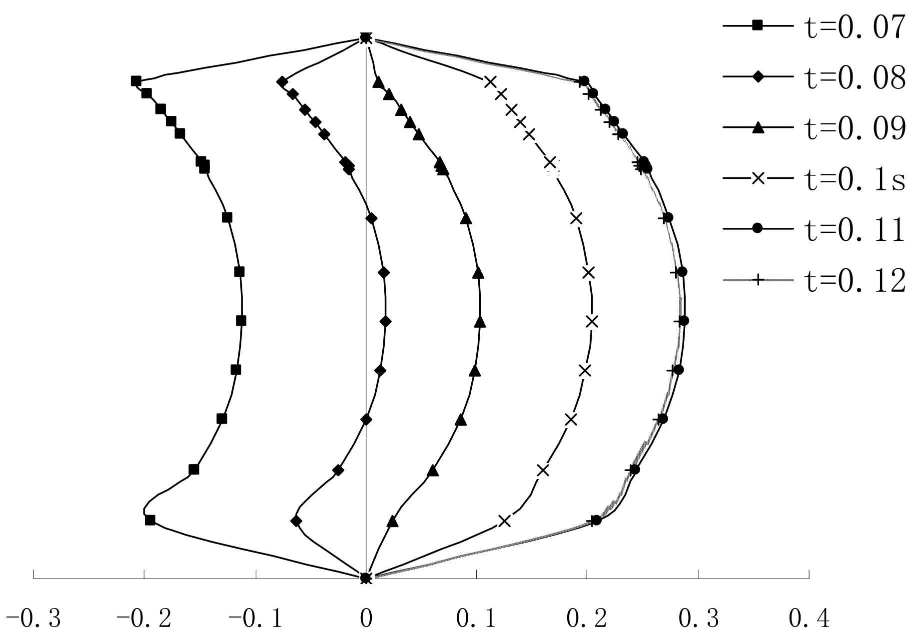

Figure 5: velocity ditribution in cross-section from 0.07s to 0.12s

The interesting thing is that, during the period of water hammer wave propagating, the velocity gradient in the code region did not varied much. While the significant variation happened near the wall, there is intensive gradient variation of water velocity. This phenomenon deserves further study.

According to the CFD results, the SIMPLE series algorithm for compressible water flow is appropriate to simulate the water hammer. And the selected turbulence model and boundary condition is also suitable.

5 NUMERICAL SIMULATION OF WATER HAMMER WHEN VALVE FAST CLOSE

5.1 Calculation Condition

Figure 6 shows a water hammer model, consisting of two tanks, a horizontal pipe, and a fast closing valve at the end of the pipe. The pipe length is 600m, with diameter 1m. The numerical method is three-dimensional CFD and one-dimensional MOC (Method of Characteristics). The results will be compared to verify the validation of the CFD simulation of water hammer.

The initial condition is that, the upstream pressure is 150m, and the downstream pressure is about 143.45m. The acceleration of gravity is 9.806m/s^2 and the velocity of the water hammer wave is 1200m/s. During the simulation, the friction is ignored, that to say, taking the water as an inviscid flow.

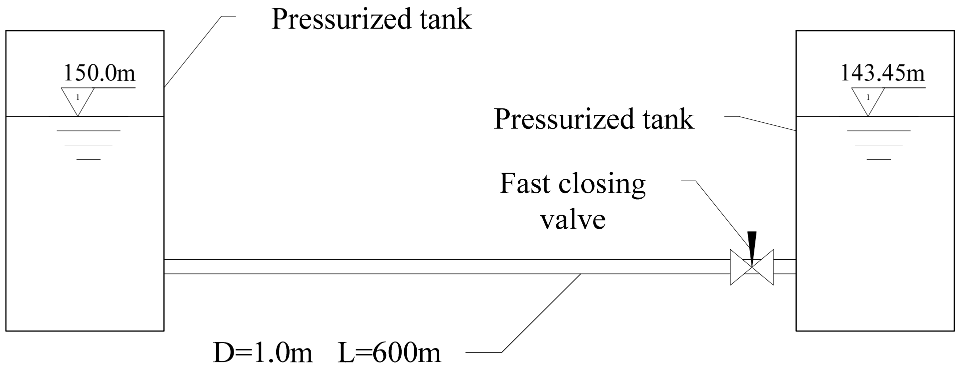

Figure 6: Computational geometry for water hammer in pipe when valve fast close

Boundary condition for CFD numerical simulation as follows, pipe inlet pressure takes a value of $p_u = \rho g HGL_u$ in steady and unsteady pipe flow. Pipe outlet pressure takes a value of $p_d = \rho g HGL_d$ in steady state, and adjusted to velocity outlet boundary and set $v = 0m/s$. The pipe applied non-slip wall boundary and the time step takes a value of $\Delta t = 0.001s$.

5.2 Results and Analysis

(1) The pressure fluctuation at the outlet and the velocity fluctuation at the inlet

Figure 7 shows the pressure fluctuation at the outlet of pipe. Figure 8 shows the fluctuation of average cross-section velocity at the inlet. The results of two approaches, not only the pressure but also the velocity fluctuation process, have a high coincidence with each other. It is concluded that, the compressible water flow of SIMPLE series algorithm is suitable for water hammer numerical simulation.

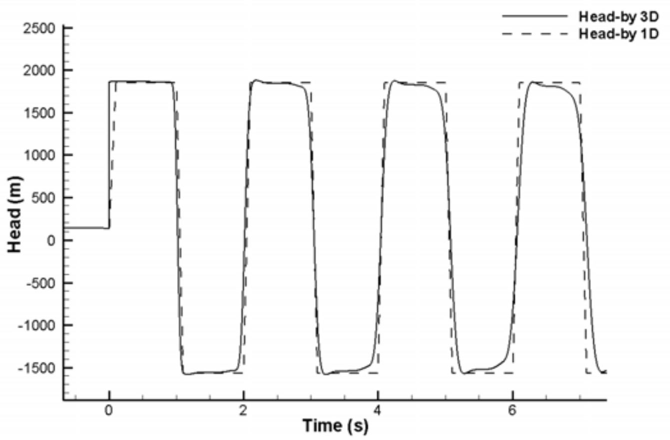

Figure 7: pressure fluctuations at the valve(outlet)

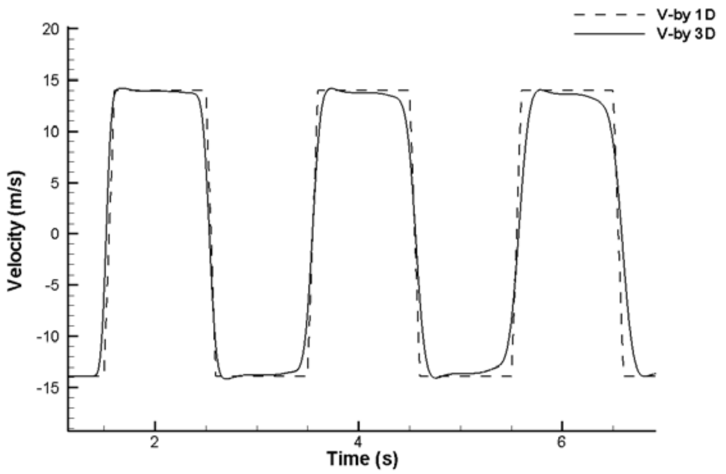

Figure 8: Water velocity fluctuations at the inlet

From the pressure fluctuation at the outlet of pipe, we can know that, the maximum and minimum water hammer pressure of one-dimension and three-dimension is very consistent at the first propagation stage of water hammer wave. But with the time increasing (the water hammer is propagating), there has occurred a slight distortion when pressure fluctuating pulse maintain at maximum and minimum water hammer pressure. As mentioned above, we think that the main cause of the distortion is the numerical dissipation. So, when the CFD computational time step being less than $10^{-2} \sim 10^{-3}$ of the water hammer propagating period, the numerical dissipation has a slight effect on numerical results.

(2) The velocity fluctuation at the middle cross section of the pipe

In order to understand the unsteady character of velocity distribution during the inviscid water hammer wave propagation, one vertical diameter of the cross section at the middle of the pipe has chosen to illustrate the numerical results. As the same as we predicted, three-dimension results shows that the velocity distribution of cross section is uniform during the inviscid water hammer wave propagation. The main reason is that we didn’t consider water viscosity and wall roughness. Figure 9 shows the velocity fluctuation in cross-section from 0.0s to 2.5s, when the velocity directed to downstream, the value is positive.

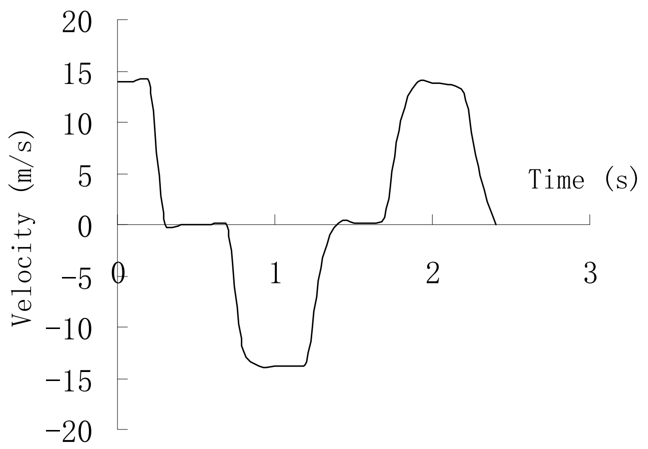

Figure 9: Averaged velocity fluctuation in cross-section from 0.0s to 2.5s

According to theoretical analysis, during one period, the water hammer will propagate through the middle cross-section four times. The first time is 0.25s, second time is 0.75s, third time is 1.25s, and fourth time is 1.75s, and so on.

When t<0.25s, the first positive wave has not reach the middle section, so the velocity is the same to the initial status.

When t=0.25s, 0.75s, 1.25s, 1.75s, the wave is penetrating the middle section, so the velocity in cross-section should be 0m/s during this time.

When t=1.00s, 2.00s, the negative or positive wave has past the middle section, so the velocity is minimum or maximum.

6 NUMERICAL SIMULATION OF WATER HAMMER WHEN VALVE GRADUALLY CLOSE

6.1 Calculation Condition

Figure 10 shows a water hammer model, consisting of two tanks, a horizontal pipe, and a gradually closing valve at the end of the pipe. The pipe length is 600m, with diameter 1m. The numerical method is 3D CFD and 1D MOC. The results will be compared to verify the validation of the CFD simulation of water hammer. The water head of upper tank is 150m, and the water head of downstream tank is 140m.

The close law of valve is shown as:

$$\tau = \left(1 - \frac{t}{t_c}\right)^{Em} \tag{5}$$

Where, $t_c = 2.1s$ is the time that valve closed, $\tau_0 = 1.0$ is the initial opening, $Em = 1.5$.

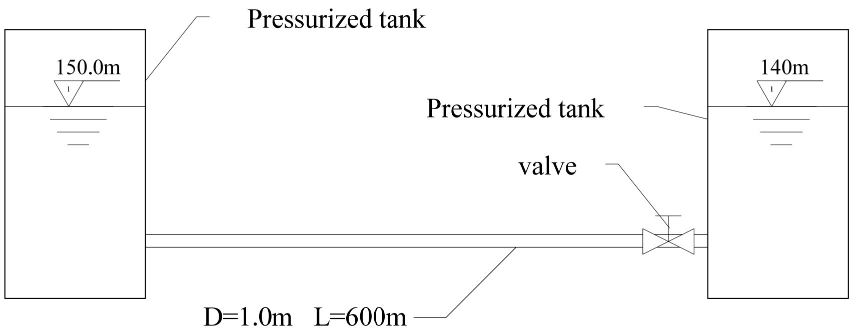

Figure 10: Computational geometry for water hammer in pipe when valve close gradually

6.2 Numerical Simulation

During the calculation of steady flow or transient flow, inlet is set as pressure boundary condition. Outlet is also set as Pressure boundary condition in steady flow. In the purpose of simulating water hammer, change outlet into velocity boundary condition. The velocity is defined by Eq.6. For contrasting with the result of 1D MOC, we ignore the influence of flux characteristic curve of the valve, assuming the pipe wall is smooth and adopt the non-slip boundary condition.

In CFD simulation, the velocity boundary condition is abided by:

$$V^{n+1} = \tau^{n+1} \frac{Q_0}{A} \sqrt{\frac{h^n}{h_0}} \tag{6}$$

Where, V^{n+1} is the outlet velocity at n+1 time step; Q_0 is the steady flow flux at pipe outlet; h_0 is the steady flow pressure head at pipe outlet; h^n is the outlet pressure head at n+1 time step; τ^{n+1} is the valve opening at n+1 time step, calculated by Eq.5; A is the cross-sectional area of valve; $\Delta t = 0.001s$ is time-step in unsteady flow.

6.3 Results and Analysis

(1) The pressure fluctuation at the vale and the discharge fluctuation at the inlet

In this situation, the results include the 3D CFD and the 1D MOC calculations. Figure 11 shows the pressure fluctuation at the valve, the discharge variation at the inlet, and the opening variation of the closing valve. It is worth notice that the discharge magnified ten times for observation conveniently in Figure 11.

The results of two approaches, not only the pressure at end of pipe but also the discharge variation at the inlet, have a high coincidence with each other. It is concluded that, the compressible water flow of SIMPLE series algorithm is suitable for water hammer numerical simulation when valve close gradually.

From the pressure fluctuation at the outlet of pipe, we can know that, the first maximum water hammer pressure by 3D CFD is slight over the results by 1D MOC. But with the time increasing, there has not any slight distortion when pressure fluctuating. As mentioned above, we think that the numerical dissipation almost has no effect on numerical results of water hammer when valve close gradually. In order to understand the influence of the discrete scheme and CFD computational time step on the water hammer numerical simulation when valve close gradually, this deserves further study.

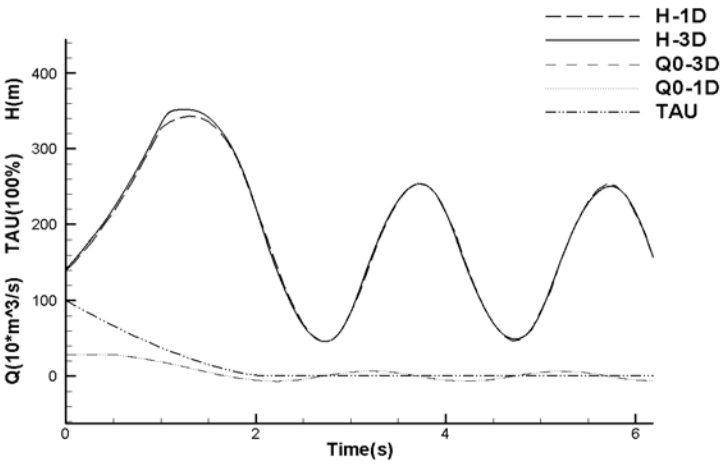

Figure 11: Comparison of the numerical simulation of water hammer by CFD and 1D MOC

(2) The distribution of velocity at the middle cross section of the pipe

In order to understand the unsteady character of velocity distribution during the water hammer wave propagation, one vertical diameter of the cross section at the middle of the pipe has chosen to illustrate the numerical results. Figure 12 shows the velocity distribution from -0.1s to 2.0s. And Figure 13 shows the velocity distribution from 0.07s to 0.12s. In the figures, when the velocity directed to downstream, the value is positive.

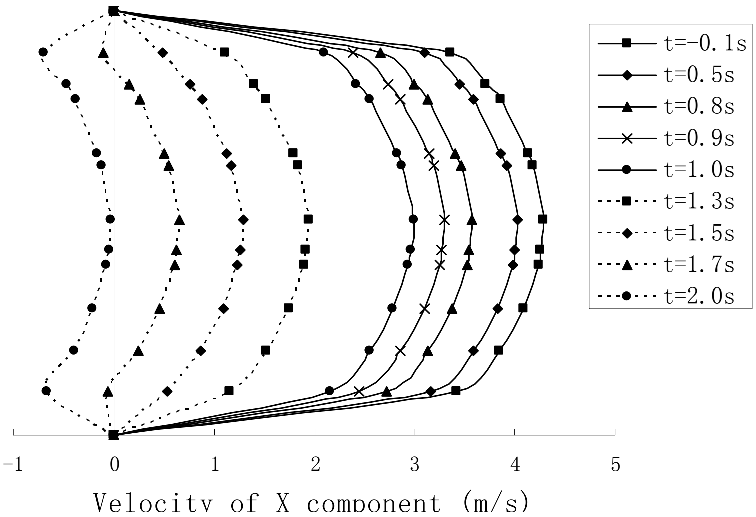

Figure 12: Velocity distribution in cross-section from -0.1s to 2.0s

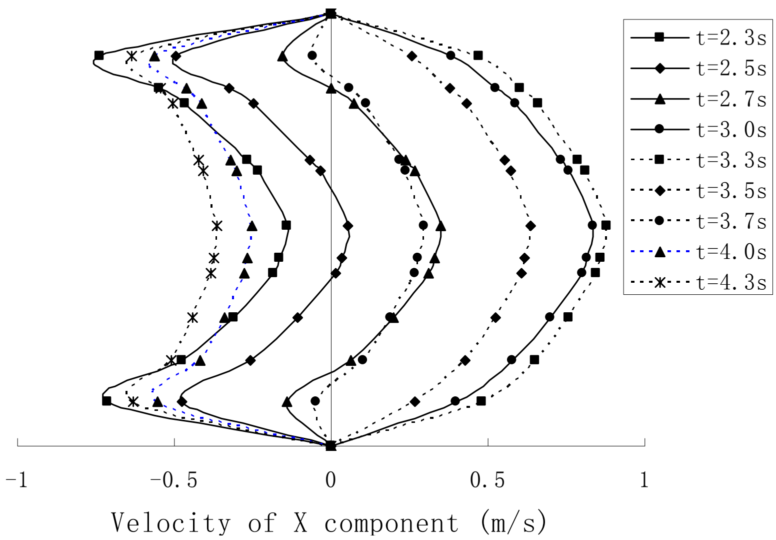

Figure 13: Velocity distribution in cross-section from 2.3s to 4.3s

As the valve opening continuous reduces, the velocity in the pipe, meanwhile, decreases (Fig.13). Before the valve closes completely (t=1.7 ~ 2.0s), the pressure caused by water hammer make the water velocity reverse. Approximately at t=2.3s, the negative velocity reaches the peak. Thereafter, the flow velocity changes from the X axis negative to its positive. The greatest positive velocity occurs about at t=3.3s. Then because of the positive water hammer wave that propagates from the end of pipe, the velocity gradually decreases and the greatest negative velocity occurs approximately.

7 CONCLUSIONS

- When the water state equation is adopted, the SIMPLE algorithm for compressible flow is used to simulate the water hammer. In order to check the reliability of the algorithm, a number of numerical calculations of water hammer are taken. The results shows that the SIMPLE algorithm for compressible flow can effectively simulate water hammer
- In numerical calculations of the "water hammer" phenomenon, the accuracy of results is affected weakly by convection discrete format, transient discrete format, interface pressure interpolation format of the series of SIMPLE algorithm, namely a numerical format with general accuracy can simulate the progress of water hammer.
- The numerical dissipation can smooth the wave front of water hammer. So, the reasonable numerical format should be selected, and the step of time should be controlled in $10^{-2} \sim 10^{-3}$ of the period of water hammer propagation.
- Through the numerical simulation of CFD, it can make a detailed instruction about the velocity profile in the whole pipe flow during water hammer. What's more, it provides reference for studying the flow features during water hammer.
- There is an interesting thing: during the period of water hammer wave propagating, the velocity gradient in the code region did not varied much. While the significant variation happened near the wall, there is intensive gradient variation of water velocity. This phenomenon deserves further study.

REFERENCES

[1] Liu Xining. A Three-Dimensionality Numerical Analysis of Turbulent Flow by SIMPLE Method. *Xi'an: Northwestern Polytechnical University Master's thesis*. (2005)

[2] Tao Wenquan. The Morden Development of Numerical Heat Transfer. *Beijing: Science Press*. (2000)

[3] Van Doormal J P , Raithby G D , McDonald B H. The segregated approach to predicting viscous compressible flows. *ASME J Turbomachinery*, pp. 265-277 (1987)

[4] Karki K C , Patankar S V. Pressure based calculation procedure for viscous flows at all speed in arbitrary configuration. *AIAA J*,. 27(9), pp. 1167-1174 (1989)

[5] Shyy W . Chen M-H, Sun C-S. Pressure-based multigrid algorithm for flow at all speeds. *AIAA J.*, 30, pp. 2660-2669 (1992)

[6] Date A W. Solution of Navier-Stokes equations on nonstagered grid of all speeds. *Numer Heat Transfer, Part B*,. 33, pp. 451-467 (1998)

[7] Zhang Moujin, Wong C.M., M.J. Pomfret. A SIMPLE Algorithm for Viscous Compressible Flows. *J. Aerospace Power*. **1**, pp.46-50 (1997)

[8] Bergant A, Simpsion A R. Estimating Unsteady Friction in Transient Cavitating Pipe Flow. *Pro. 2nd Int. Conf. On water pipeline system*, Edingburgh, UK,24-26 May 1994, BHRA Group Conf. Series Publ. No. 110, pp. 3-15 (1994)