Numerical Modeling of the Water Hammer Phenomena With FLUENT Software

Abolfazl Shamsai[i]; Hamidreza Rabie Far[ii]; Seyed Taha Alavi[iii]

ABSTRACT

Water-hammer phenomenon occurs in pressure pipes due to sudden change in boundary conditions such as a quick opening and closing of the valves, sudden connection and disconnection of a pump or turbine and similar situations. In this case, as a result of sudden changes in the momentum of the flow, high speed pressure wave flows through the pipelines and if the maximum pressure created goes beyond the maximum pressure allowed for the system, its destruction is inevitable. Therefore it is essential to consider these factors during designing of hydraulic systems.

In this study the water hammer phenomenon is simulated with the help of FLUENT software and implementation of UDF s(user defined functions) in the program to consider different aspects such as wall and fluid elasticity. The results of this modeling process were verified by Araya Experiment for water-hammer and after the verifications, the data obtained from the model were used to analyze the water-hammer phenomenon in series of pipes and their effect on the severity of the water-hammer impact. It has been derived for series of pipes, combination of reflected pressure with initial pressure in pipe junction creates waves with higher pressure head and lower frequency. These effects are intensified especially when origin of water hammer is in pipe with larger section. FLUENT can be used as a tool for initial estimation of water hammer phenomenon in series of pipes, and obtaining value of primary head in pipe systems.

KEYWORDS

Water hammer, FLUENT, unsteady friction coefficient, series pipes.

1. INTRODUCTION

The temporary condition caused in fluid transport systems where the wall's and fluid's elasticity can not be overlooked is called water hammer phenomenon. This usually occurs in liquid transmission pipelines and at the time of valves opening or closing, starting or disconnecting pumps or changing of the turbine's load. The changes in momentum that result from the above mentioned factors move through out the system in the form of pressure waves and with high speed and if their maximum pressure passes the level allowed in the system, it could cause damage or a total destruction of the system. Therefore during design stage of any hydraulic systems; this phenomenon should be taken into consideration.

In initial models used for simulating water-hammer phenomenon, steady friction coefficients like Darcey-Weisbach and Moody diagram were frequently used in equations. Today these models are labeled as the "classic" water-hammer models [11].

For those flows in which the regional changes in Reynolds factor are not considerable, it is helpful to assume the steady and unsteady friction coefficient equal. But in water hammer phenomenon and in the classic model such assumptions result in under-estimation of friction coefficient and therefore the pressure fluctuations are estimated higher than the real situation [1]. In other words, this model could only calculate the primary maximum pressure of water hammer phenomenon correctly.

In the past two decades, different models were presented to calculate the unsteady friction coefficient [2,3]. These models have been relatively successful in determining the tension and fluctuations of water hammer phenomenon. Among those are Zielke's model [4] for steady flow and Brunone's model [5] [6] for turbulent (chaotic) flow. These models include terms that are added to the steady friction coefficient to simulate the unsteady condition of friction coefficient.

Today methods for analyzing the water hammer

[i] Professor,Science and Research Branch,Islamic Azad University,Tehran,Iran,Email: Shamsai@Sharif.edu
[ii] PhD student,Science and Research Branch,Islamic Azad University,Tehran,Iran,Email: Rabiefar@yahoo.com
[iii] MS student,Science and Research Branch,Islamic Azad University,Tehran,Iran,Talavi@yahoo.com

phenomenon are mostly based on using "method of characteristics" together with "unsteady friction approximation" , and the resulting equations are solved with finite differences methods[10,11,12,13].

However, in present study, FLUENT software was used to model the water hammer phenomenon. This should be mentioned here that FLUENT does not have any option for unsteady friction approximations. The effects of friction in pipes lead to creation and growing boundary layers between the wall and the fluid and make a change in velocity profile between pipe input and output. In FLUENT, to make a correct model for boundary layer, certain experimental correlations are used, these models simulate the velocity profile near the walls correctly. Considering the points mentioned so far, whether FLUENT could model properly water-hammer phenomenon or not is a subject for current investigation.

2. Equations Involved in Water Hammer Phenomenon

Using momentum and mass conservation equations along the pipeline, the following equation can be derived:

$$\frac{dV}{dt}+\frac{1}{\rho}\frac{\partial p}{\partial s}+g\frac{dz}{ds}+\frac{f}{2D}V|V|=0 \quad (1)$$

$$a^2\frac{\partial V}{\partial s}+\frac{1}{\rho}\frac{dp}{dt}=0 \quad (2)$$

These two are the main water hammer equations. In equation (2), a is the speed of pressure wave and considering the interactions between the wall and fluid we would have:

$$a=\frac{\sqrt{K/\rho}}{\sqrt{1+\frac{K}{E}\frac{D}{e}\left(1-\mu^2\right)}} \quad (3)$$

If the friction coefficient, in equation (2), is calculated by Darcey-Weisbach equation, the new equation will be the classic water hammer equation.

To solve this problem, the unsteady friction coefficient methods were developed. In Brannon's model [5] of unsteady friction coefficient, the coefficient term is a function of average momentary velocity, V, acceleration, $\frac{\partial V}{\partial t}$, average momentary acceleration، $V\frac{\partial V}{\partial x}$ .Like most unsteady friction coefficient methods, the friction coefficient consists of two parts: the Darcey-Weisbach semi-steady part, f_q, and the unsteady part ، f_u where:

$$f=f_q+f_u \quad (4)$$

At the end, according to Brunone's model [5] the unsteady friction coefficient is defined as :

$$f_u=\frac{kD}{V|V|}\left(\frac{\partial V}{\partial t}-a\frac{\partial V}{\partial x}\right) \quad (5)$$

3. Equations in FLUENT

FLUENT [7] is general purpose CFD software in which according to boundary conditions and primary conditions of the conservation equations of mass, momentum and energy are solved using the finite volume method for the considered geometry. These equations have the following general form:

$$\frac{\partial}{\partial t}\int_V \rho\varphi dV+\oint_A \rho\varphi V.dA=\oint_A \Gamma\nabla\varphi.dA+\int_V S_\varphi dV \quad (6)$$

Unsteady Convection Diffusion Generation

By changing φ, as showed in Table (1), general conservation equations will be derived.

Table (1)

Equation	φ
Continuity	1
X-momentum	u
Y-momentum	v
Energy	e

FLUENT provides the following choices of turbulence models: Spalart-Allmaras model, k-ϵ models, k-ω models, Reynolds stress model (RSM), Detached eddy simulation (DES) model, Large eddy simulation (LES) model. It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation.

The simplest complete models of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k- ϵ model in FLUENT falls within this class of turbulence model and is used in the current paper. In FLUENT the turbulence kinetic energy, k, and its rate of dissipation, ϵ, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)= \frac{\partial}{\partial x_j}\left[(\mu+\frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \quad (7)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)= \frac{\partial}{\partial x_j}\left[(\mu+\frac{\mu_t}{\sigma_\varepsilon})\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \quad (8)$$

There are two methods available to correctly model the field near the walls in FLUENT, named as wall function and near wall model.

In wall function method, the flow field is not solved near the walls and the velocity profile for this area is calculated by semi empirical correlation. Boundary layer between the fluid and walls is divided into several parts and in each area the velocity profile is calculated by the semi empirical correlation valid for every part. These areas are demonstrated in figure (a-1)

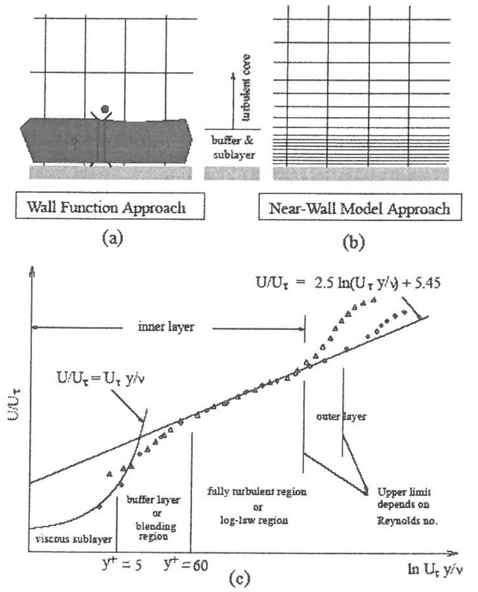

Figure (1) Two methods of analyzing border layer (a) and (b) and the semi empirical correlation in wall friction (c) [7].

The characteristics of wall friction in brief are:

-the border layer is modeled by semi empirical correlation.

- Without solving this area, the software connects it directly to the turbulent area by bridging over it.

- In turbulent model equations terms are inserted to include the effect of wall presence.

- In fluids with high Reynolds factor this method reduces the time needed for solving the problem, considerably.

- In industrial applications as a result of tendencies towards using the extremely turbulent (chaotic) flows, this method is even more precise and economic.

- But for flows with lower Reynolds factor, since the entire boundary layers are not developed, this model is not exact and reliable. In such situation it is better to solve the boundary layers with the whole field of pipe.

Another method for treating boundary layers is the near wall function method, in which by choosing the small mesh near the wall, the method practically models the boundary layers and solves it.

4. Verification

4.1 Araya Test

Water hammer phenomenon for simple pipe flow was conducted by Araya [8]. In 1993, Araya prepared the following equipment for this experiment in Albrook hydraulic laboratory of Washington University.

Table (2) Experiments characteristics

parameter	Primary value
Primary discharge	0.0165 m^3/s
Water-level in tank	2.65 m
Friction coefficient	0.0223
Primary valve location	0.62
Wave speed	1125 m/s
Water density	999.1 kg/m^3
Dynamic viscosity	1.06×10^{-3} m^2/s

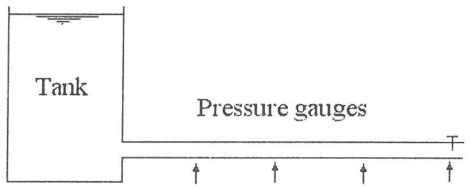

Figure (2) Araya experiment design and the related data.

The experiment involves an open tank that has a steel pipe with the inside diameter of 0.101 m , thickness of 0.00635 m and length of 32m attached to it. A petcock tap is placed at the end of the pipe. The water that exits the system through the pipe is gathered in an in-between container and is pumped back into the main tank, so that the water level stays the same through out the experiment. Four pressure gauges are placed on the 0.25 L, 0.5 L, 0.75 L and L of the pipe. In stable conditions water exits the pipe with a discharge of 0.0165 m^3/s which is equal to Reynolds figure (2.06×10^5).

Transient phase in the system begins with closing the tap gradually. At first only 0.62 parts of the tap are open and after that it is completely closed in a time span of 1.085 seconds. The pressure fluctuations caused in the system are recorded for the next two seconds after the beginning of the temporary phase.

In FLUENT the sliding mesh method was used to model the closing of the tap. The relation between wave speed and fluid compressibility (based on Bulk Modulus) were defined to the software as UDF (user defined functions) programs in C language. And wall function method was used for simulation of boundary layers.

The pressure fluctuations caused by closing the tap, where it is placed in the middle of the pipe from point 4 are displayed in Figures (3-A) and (3-B). The straight lines show the experimental data and the dotted lines are indicator of the data obtained from the classic water hammer theory and are calculated with the assumption that friction coefficient is fixed. The FLUENT results are shown in Figures (4-A) and (4-B).

As we can see, the FLUENT results are more desirable than the classic water-hammer data, but the pressure wave in FLUENT fades a little faster than from the real situation. This indicates that the friction coefficient for water hammer in wall function model would be estited higmaher than the real situation.

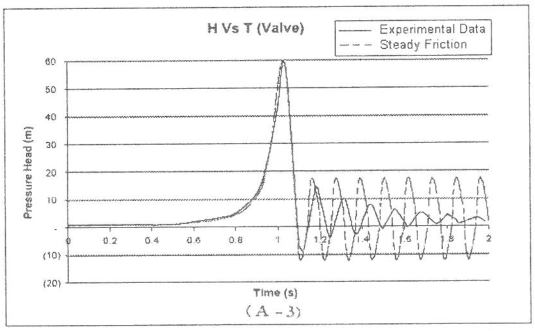

Figure (3-A) a comparison of Araya experimental data and classic water hammer theory data (History of static pressure near valve).

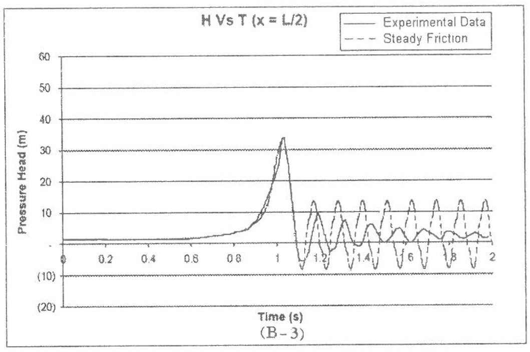

Figure (3-B) a comparison of Araya experimental data and classic water hammer theory data (History of static pressure in L/2 of pipe).

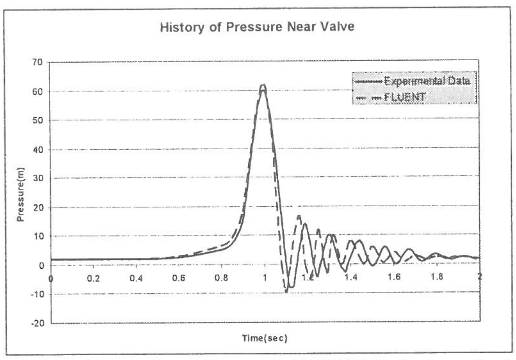

Figure (4-A) a comparison of FLUENT data and Araya experimental data (History of static pressure near valve).

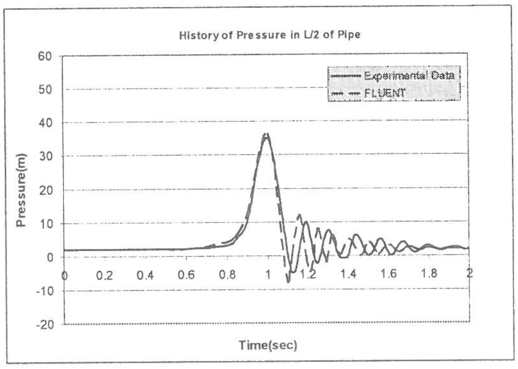

Figure (4-B) a comparison of FLUENT data and Araya experimental data (History of static pressure in L/2 of pipe).

4.2 Bergant Test

In this test, the effect of unsteady friction on water hammer wave forms is investigated for the case of rapid closure of the downstream end valve(Fig. 5). The sloping pipe is 37.2 m long with internal diameter of 22 mm and 1.6 mm of wall thickness. The experimental apparatus is fully described by Bergant et al. [14].

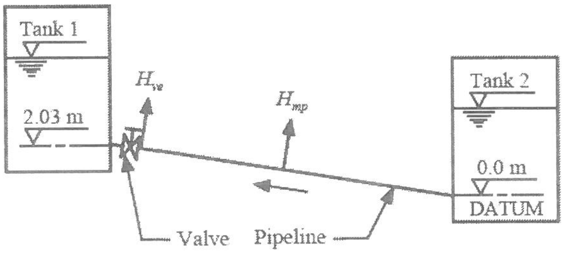

Figure (5) Experimental Apparatus of Bergant.

Experimental tests, starts with the initial flow velocity of 0.2 m/s. In tank 2, the static head is 32 m.Valve closure time and water hammer wave speed are 0.009 s, and 1319 m/s ,respectively [14]. Fluent and Experimental results are compared in Table3 and Fig. 6.

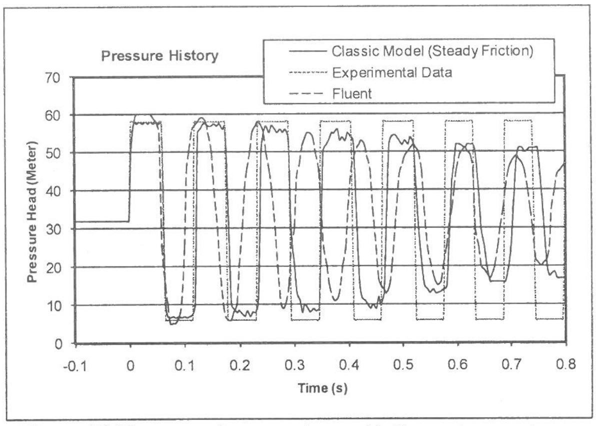

Figure (6) Fluent result comparison with Bergant apparatus.

Table (3) Results of comparison

parameter	Experimental	Fluent
Max Initial Head (meter of H2o)	58	59.1
No of Oscillations In First 0.8 sec	7	8.5

Like previous section, pressure wave in FLUENT fades a little faster than from the real situation thus, the number of oscillations calculated by Fluent is more than those counted in the real experiment.

5. WATER HAMMER PHENOMENON IN SERIES OF PIPES

The water hammer phenomenon in series of pipes has a more complicated form. Mitosek et al [9] carried out some experimental investigation in this field. Their work shows that pressure wave produced by water hammer moves through one pipe and in junctions of two pipes divide in two parts. One part is reflected back while the rest of the pressure wave moves through another pipe. The resulting pressure wave is a combination of the initial wave and the reflected one. To show this phenomenon and investigate the effect of the type of pipe junctions on water hammer, two series steel pipes, as contraction and expansion (shown in Figure 7) were modeled by FLUENT, as we can see in second setup arrangement of two pipes have been changed. The thicker pipe is 2 meters long and the smaller pipe is half that size. The wave's speed is 1250 meters per second and the pressure equals 20 kPa. The water hammer is created by the sudden blockage of the fluid in the exiting point. The pressure fluctuation on the exiting valve point and in the middle of the pipe where the fluid enters (points shown in the figure) are recorded.

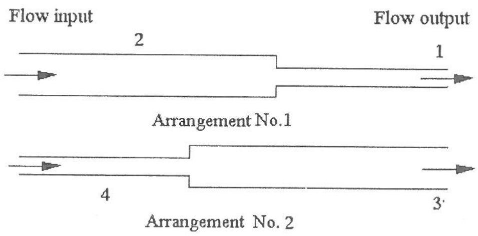

Figure (7) the arrangement design for pipes in series.

In Figure 8, the pressure fluctuations for the points shown in Figure 7 are demonstrated. As it was expected where the smaller pipe is placed near the exiting valve (contraction) the water hammer phenomenon is more intense and has bigger fluctuations.

To analyze the effect of contraction area on maximum pressure head of water hammer, five arrangements with different D/D_o were used. D and D_o are the pipe diameters before and after contraction.

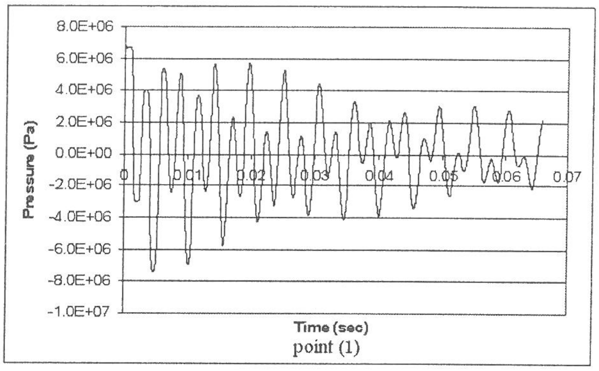

Figure (8-1) pressure fluctuations for point 1.

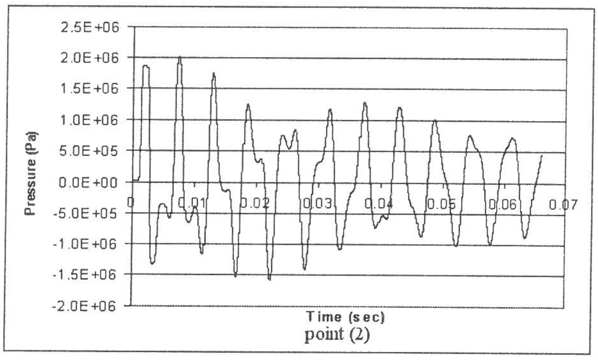

Figure (8-2) pressure fluctuations for point 2.

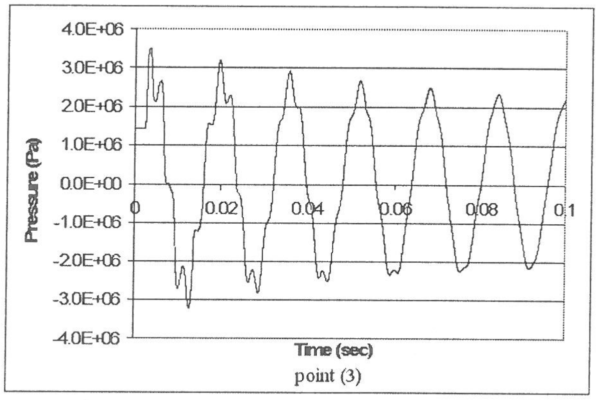

Figure (8-3) pressure fluctuations for point 3.

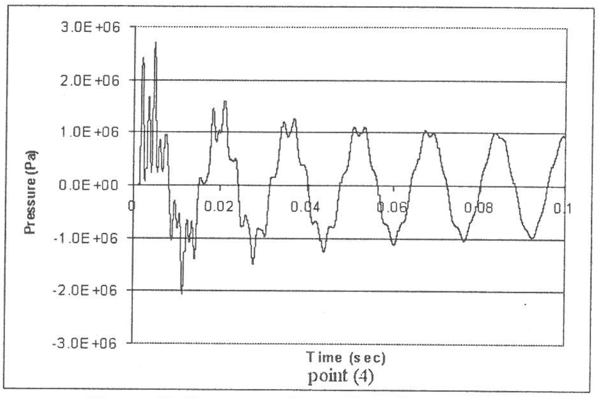

Figure (8-4) pressure fluctuations for point 4.

Table (4) summaries the results of these calculations. In the first arrangement, D/D∘ of one denotes that there is no contraction in flow path. Maximum pressure head for this arrangement is P∘= 8 Mpa.

The following table shows that as the ration of D/D∘ decreases, the maximum water hammer head will decrease too, but the number of oscillations will increase. Another calculation has been done to determine the effect of pressure head on water hammer in series of pipes. Increasing input pressure head for arrangements of Fig. 5 from 20kPa to 40kPa causes that maximum pressure increases 1.36 times for contraction and 1.28 times for expansion. These analyses show the effect of input head on water hammer maximum pressure head.

Table (4) Effect of contraction on water hammer maximum head

D/D∘	Maximum Head	No of Oscillation
1	P∘	15
0.8	0.75 P∘	15
0.6	0.6 P∘	16
0.4	0.55 P∘	17
0.2	0.53 P∘	18

6. Conclusion

- The modeling of the water hammer phenomenon with the help of FLUENT software was investigated in this study. The points that should be taken into consideration in this modeling is that this software does not have separate equations or method for calculation of unsteady friction. But with the help of semi empirical correlation used in boundary layer modeling of FLUENT, proper simulation of water hammer was derived. To make a better modeling of water-hammer phenomenon, the physics of the experiment and turbulence flow reviewed and proper turbulence model is selected for simulation.
- Although in initial time of starting water hammer phenomenon these models are not much different from each other, after some pressure oscillations, differences in their results can not be overlooked.
- In smaller geometries the results obtained from this software are in acceptable range according to real data but for larger size systems, FLUENT results are to some extent less than the real data and disagreement appeared gradually between FLUENT results and experimental data. In large geometries, effect of unsteady friction become more important and the lack of proper unsteady model in FLUENT can be one reason of this behavior.
- Pressure reflection in junction of series pipe made the behavior of water hammer more complicated in series pipe with respect of single pipes. Combination of reflected pressure with initial pressure creates waves with higher pressure pick and lower frequency. Both of these two factors are undesirable and have destructive effect on hydraulic systems.
- When origin of water hammer is in pipe with lower cross section, comparison of results (Figs. 6) shows that pressure oscillation damps in another pipe (arrangement two in Fig. 5), but when water hammer occurred in pipe with larger cross section, high oscillation pressure wave is initiated in the next pipe (arrangement one in Fig. 5).
- FLUENT can be used as a tool for initial estimation of water hammer phenomenon in series of pipes, and obtaining value of primary head in pipe systems.

7. Table of Acronyms

Q : fluid discharge

V : average momentary velocity

ρ : fluid density

E :pipe wall elasticity

e :wall coarseness

P :pressure

f_q : Darcey-Weisbach-friction coefficient

f_u : unsteady friction coefficient

k : steady non-retentive friction coefficient

K :Bulk Modulus

μ :Poisson coefficient

a :wave speed

t :time

8. References

[1] Larock, B., Jeppson, R., Watters, G., “Hydraulics of Pipeline Systems”, CRC press, 2000.

[2] Ghidaoui, M.S., Mansour, S., "Efficient Treatment of the Vardy-Brown Unsteady Shear in Pipe Transients," Journal of Hydraulic Engineering, Vol.128, No.1, pp.102-112, 2002.

[3] Bergant, A., Tijsseling, A., “Parameters Affecting Water Hammer Wave Attenuation, Shap and Timing”, 10th International Meeting of the Workshop on the Behavior of Hydraulic Machinery, June 26.-28. 2001 , Trondheim, Norway.

[4] Zielke, W., "Frequency-Dependent Friction in Transient Pipe Flow," Journal of Basic engineering, Vol.90, No.1, pp.109-115, 1968.

[5] Brunone, B., Golia, U.M., Greco, M., "Effects of Two-Dimensionality on Pipe Transients Modeling," Journal of Hydraulic Engineering, Vol.121, No.12, pp.906-912, 1995.

[6] Bergant, A., Simpsom, A.R., Vitkovsky, J.P., "Developments in Unsteady Pipe Flow Friction Modeling," Journal of Hydraulic Research, Vol. 39, No.3, pp.249-257, 2001.

[7] FLUENT 6.2 Documentation, FLUENT Inc,2005.

[8] Silva-Araya, "Energy Dissipation in Transient Flow", Ph.D. Dissertation, Washington State University, Washington, 1993.
[9] M. Mitosek, M. Chorzelski, A. Malesińska," Experimental Analysis of Natural Frequency of Water Column Due to Water Hammer in Series Pipe Systems". 5th International Conference on Hydro -Science & -Engineering (ICHE-2002), Warsaw.
[10] Larock, B., Jeppson, R., Watters, G., "Hydraulics of Pipeline Systems", CRC press, 2000.
[11] Pezzinga, G., "Quasi-2D Model for Unsteady Flow in pipe Networks," Journal of Hydraulic Engineering, Vol. 125, No.7, pp. 676-685, 1999.
[12] Suzuki, K., Taketomi, T., Sato, S., "Improved Zielke's Method of Simulating Frequncy-Dependent Friction in Laminar Pipe Flow," Journal of Fluid Engineering, Vol.113, No.4, pp.569-573, 1991
[13] Zielke, W., "Frequency-Dependent Friction in Transient Pipe Flow," Journal of Basic engineering, Vol.90, No.1, pp.109-115, 1968.
[14] Bergant, A., Simpson, A.R., and VstkovskÁ, J., Review of unsteady friction models in transient pipe flow. Proceedings of the 9th International Meeting of the IAHR Work Group on the Behaviour of Hydraulic Machinery under Steady Oscillatory Conditions, Brno, Czech Republic (1999).