11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

Computations of Laminar and Turbulent Water Hammer Flows

S. D. Saemi[1*], M. Raisee[2], M. J. Cervantes[3] and A. Nourbakhsh[4]

1. School of Mechanical Engineering, College of Engineering, University of Tehran, P. O. Box: 11155-4563, Tehran, Iran, si.saemi@ut.ac.ir
2. School of Mechanical Engineering, College of Engineering, University of Tehran, P. O. Box: 11155-4563, Tehran, Iran, mraisee@ut.ac.ir
3. Division of Fluid and Experimental Mechanics, Lulea University of Technology, SE 971 87, Lulea, Sweden. Michel.Cervantes@ltu.se
4. Hydraulic Machinery Research Institute, School of Mechanical Engineering, College of Engineering, University of Tehran, P. O. Box: 11155-4563, Tehran, Iran, anour@ut.ac.ir

Key Words: *Water hammer, laminar flow, turbulent flow, numerical, validation*

In this paper numerical results for both laminar and turbulent water hammer flows in circular pipes are presented and discussed (see Figure 1).

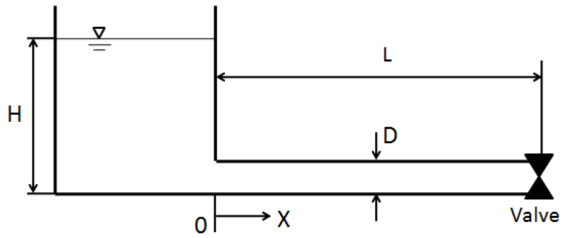

Fig. 1: Pipe geometry

The term 'water hammer' is used to describe the unsteady fluid flow behavior when a fluid in motion is forced to stop in a closed system such as pipe network. This phenomenon produces intensive pressure waves that travel periodically along the pipe. One of the main issues in water hammer prediction is the application of reliable unsteady computational models for the predictions. The water hammer phenomenon has been simulated numerically by use of one-dimensional and two-dimensional models. On-dimensional models are based on friction correlations while two-dimensional models use turbulence models. In 2D simulations, the most widely used turbulence models in the literature are algebraic turbulence models because of their simple mathematical formulation. More recently, Riasi et al. [1] used the Wilcox k-ω model to study water hammer in a straight pipe. All of the studies reported in the literature relied on simplified water hammer equations for the predictions and did not use full governing equations. In this work, for the first time, we investigated the water hammer in a straight pipe based on Reynolds averaged Navier-Stokes (RANS) equations using finite volume method. The CFD-based simulations of water hammer are performed using FLUENT software. The RNG k-ε turbulence model with wall functions and the k-ω SST turbulence model are used for turbulence modeling. Results of both test cases are compared with the available experimental data in the literature. To account for the effect of pipe elasticity a new Bulk modulus of elasticity $K_f' = K_f / (1 + K_f D / eE)$ is introduced, because a rigid pipe assumption will give rise to a change of pressure larger than for an elastic one leading to a higher wave speed which is incompatible with the experimental data. Introducing a new Bulk modulus based on the change of pipe diameter is appropriate to solve the water hammer phenomenon. The water hammer wave speed is thus given by:

$$a = \sqrt{\frac{K_f/\rho}{1 + K_f D/eE}} \quad (1)$$

where K_f, E, e and D are the bulk modulus of elasticity of the fluid, the Young modulus of elasticity, the thickness of the pipe and the diameter of the pipe respectively. The flow is assumed to be compressible and isothermal. Thus, the density variation depends only on the pressure change and is computed based on the following equation:

$$d\rho = \rho dp / K'_f \quad (2)$$

This equation provides the relationship between the pressure and the density change and implemented by a User Defined Function (UDF) into the FLUENT software.
Fig. 2 shows the predicted head verses time compared with the experimental data of Holmboe et al [4]. It can be seen that the computed distribution of head at the mid-point between inlet and outlet of the pipe is in excellent agreement with the experimental data.

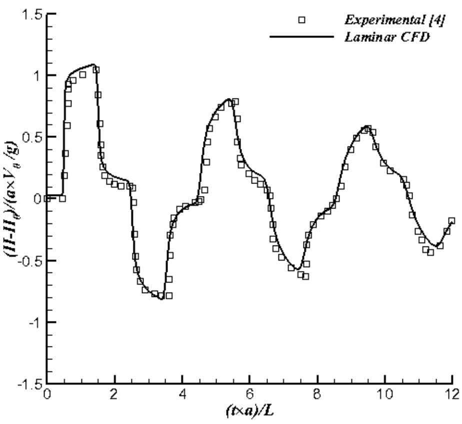

Fig. 2: Head of the flow at the mid-section as a function of time, Experimental: Experimental results of Holmboe et al. [4] , L: pipe length, a: speed velocity, V_0: initial water velocity

Furthermore comparisons for the turbulent case showed that the RNG k-ε model somewhat overpredicts the head variation and low-Re k-ω SST turbulence model can accurately predict the pressure and velocity profiles caused by water hammer.
In the proposed paper results, including the head change at the pipe midsection as a function of time, the velocity profile at the midsection as a function of time and the shear stress variation on the wall, for both test cases will be presented and discussed.

References

[1] Riasi, A., Nourbakhsh, A., Raisee, M., 2009, Unsteady turbulent pipe flow due to water hammer using k-ω turbulence model, J. of Fluids Engineering, 47(4), pp. 429-437.
[2] Wylie, E. B., Streeter, V. L., and Suo, Lisheng, 1993, Fluid Transient in Systems, Prentice-Hall, Englewood Cliffs.
[3] Guinot, V., 2002, Riemann Solvers for Water Hammer Simulations by Godunov Method, Int. J. Numer. Methods Eng., 49, pp. 851–870.
[4] Holmboe, EL., Rouleau, WT., 1967, The effect of viscous shear on transients in liquid lines, Journal of Basic Engineering, Transactions of the ASME, 89(1):174–180.