Water Hammer Simulation by Implicit Finite Difference Scheme Using Non-Symmetrical Staggered Grid

MORTEZA SEPEHRAN¹, MOHAMMAD ALI BADRI NOUDEH²
¹Amir Kabir University of Technology
Chemical Engineering Department
No. 24 Hafez Ave. Tehran
IRAN
msepehran@pogc.ir , http://www.aut.ac.ir/
²Islamic Azad University, Abhar Branch
Vahdat Blvd, Abhar
IRAN
badri@abhariau.ac.ir , http://iau-abhar.iau.ofis.ir/

Abstract: - In this paper, Implicit Finite Difference scheme using staggered grid is applied for solving water hammer equations. The staggered grid is constructed non-symmetrically to facilitate accurate implementation of boundary conditions. The model is applied to a classic problem (systems consisting of a reservoir, a pipe and a valve). In this approach the boundary conditions such as reservoir with pipe entrance loss and valve are implemented by definition of pseudo-nodes at the grid boundaries. The comparison between experimental data and numerical results shows that the required damping and smoothing of the pressure wave can be obtained by using the staggered grid. The first maximum of the pressure waves is computed in close agreement with MOC approach.

Key-Words: -Water Hammer, Numerical Solution, Staggered Grid, Finite Difference, Transient Flow, Hydraulic

1 Introduction

Water hammer is a pressure surge or wave resulting when a liquid in motion is forced to stop or change direction suddenly (momentum change). Water hammer commonly occurs when a valve is closed suddenly at an end of a pipeline system, and a pressure wave propagates in the pipe. It may also be known as hydraulic shock. This pressure wave can cause major problems, from noise and vibration to pipe collapse.

The basic theory of water hammer has been developed by Joukowski in 1897. He derived a formula for the wave velocity, taking into consideration the elasticity of both water and pipe walls. However the general theory of water hammer was developed in 1902 by Allievi [1]. Allievi also produced charts for pressure rise at a valve due to uniform valve closure [2]. Further refinements to the governing equations of water hammer appeared in Jaeger, Wood, Rich, Parmakian, Streeter, Lai and Wylie [2]. Their combined efforts have resulted in the classical mass and momentum equations for one-dimensional water-hammer flow which is usually the basis for the numerical simulation of water hammer events.

Various numerical approaches have been introduced for pipeline transient calculations. They include the method of characteristics (MOC), finite difference (FD), wave plan, and finite volume (FV). Among these methods, the method of characteristics is the preferred method of solution. In this method the partial differential equations transform into ordinary differential equations along characteristics line and the conventional approach is to define a fixed grid in the distance-time plane. On this grid, the unknown pressures and velocities (or heads and discharges) are numerically computed in a time-marching procedure that starts from a given initial condition [1], [3],[4].

MOC method calculates an accurate prediction of the maximum pressure in the system which usually occurs during the first pressure peak. It also correctly predicts wave periods, but it usually fails in accurately calculating damping and dispersion of wave fronts [5]. The experiment shows that the pressure oscillations are relatively quickly damped and smoothed and after some second they practically
disappear. Conversely, in the results of calculation, such strong damping is not observed [6].

Unfortunately, such a discrepancy between the calculations and the experimental results is reported by many authors [6], [7], they relate the discrepancies to the fact that a number of effects are not taken into account in the standard theory of water-hammer [5]. These include: dissolved and free air in the liquid, unsteady friction in the transient flow [6], non-elastic behavior of the pipe-wall material and fluid–structure interaction (FSI) [8], [9].

Generally the better agreements between the results of calculations and observations could be obtained by considering these improvements, but still the problem of discrepancy between the numerical solution of improved water hammer equations and the experimental data is reported in many papers.

This paper presents a different way of water-hammer computation. We focus our attention on modification of solution method by using the staggered grid instead of improvement of system equations [10]. The solution method is similar to the method introduced for solving the shallow water equations. For accurate implementation of boundary conditions a modified staggered grid (non-symmetrical staggered grid) has been introduced.

A relatively simple problem for which experimental data are available has been considered. The computed heads are compared with both the experimental data and the results of system equations solving by method of characteristic.

2 Governing equations

The unsteady, one-dimensional incompressible fluid flow in a closed conduit is governed by the following two hyperbolic partial differential equations, the momentum and continuity equations [1], [3].

\[
\frac{\partial H}{\partial t} + \frac{a^2}{gA} \frac{\partial Q}{\partial x} = 0 \tag{1}
\]

\[
\frac{\partial Q}{\partial t} + gA \frac{\partial H}{\partial x} + \frac{f}{2DA} |Q|Q = 0 \tag{2}
\]

Where \(Q\) = flow discharge (flowrate); \([\text{m}^3/\text{s}]\)

\(H\) = piezometric head; \([\text{m}]\)

\(a\) = wave velocity; \([\text{m/s}]\)

\(g\) = acceleration of gravity; \([\text{m}^2/\text{s}^2]\)

\(x\) = coordinate axis along the conduit length;

\(A\) = cross-sectional area of the pipe; \([\text{m}^2]\)

\(t\) = time; \([\text{s}]\)

\(D\) = internal pipe diameter; \([\text{m}]\)

\(f\) = Darcy-Weisbach friction factor;

![Fig. 1 Reservoir-pipe-valve system](image)

\(A\) = cross-sectional area of the pipe; \([\text{m}^2]\)

\(t\) = time; \([\text{s}]\)

\(D\) = internal pipe diameter; \([\text{m}]\)

\(f\) = Darcy-Weisbach friction factor;

2.1 Boundary and initial conditions

The system consists of a large reservoir at the upstream end of the pipeline and a valve at the downstream end discharging to the atmosphere.

The mentioned model with above system equations requires two boundary conditions at the ends (i.e., at \(x = 0\) and at \(x = L\)) Fig. 1.

For the reservoir at the upstream end of a piping system \((x=0)\) considering loss of entrance, the following relation can be written between head and flowrate:

\[
H_{ent} = H_{res} - \frac{(1 + k_e)}{2gA^2} Q^2 \tag{3}
\]

In which \(H_{res}\) =height of the reservoir water surface above the datum, \(k_e\) = coefficient of entrance loss, \(H_{ent}\) =head at the first point of the pipe entrance that is connected to the reservoir.

For a valve at the downstream of piping system \((x=L)\), the following relation can be written between valve opening percent %, valve flowrate and valve upstream head in both transient and steady states [1], [3]:

\[
Q_s = \frac{Q}{\sqrt{H_s}} \frac{\tau}{\sqrt{H_t}} \tag{4}
\]

Where \(H_s\), \(Q_s\) and \(H_t\), \(Q_s\) are valve flowrates and heads upstream of the valve at steady and transient state respectively. [1], [3]. Here \(\tau\) is the relative valve opening defined as:

\[
\tau = \frac{C_d A_s}{(C_d A_s)_s} \tag{5}
\]
Where $C_d$ = discharge coefficient and $A_v$ = area of valve opening. The subscript’s' indicates steady-state conditions. The volumetric flow as a function of valve stem depends on the type of valve and is specified by the manufacturer. In the simulation, therefore it is necessary to obtain the relative valve opening for the valve closure during transient conditions.

### 2.2 Computational grid

In order to obtain second order spatial accuracy in discretization and also to facilitate accurate implementation of boundary conditions, the one-dimensional staggered grid of Fig. 2 is used [10].

With constant mesh size $\Delta x$, we approximate transient flowrate $Q(t, x)$ in $x_i = \Delta x \times (i - 1)$ and transient head $H(t, x)$ in $x_{i+1/2} = \Delta x \times (i - 1/2)$, $i=1,2,\ldots,m$. Therefore the grid starts with node at defined flowrate $Q_1$ and ends with node at defined head $H_{m+1/2}$. Based on such a grid definition, the mesh size $\Delta x$ can be calculated as $\Delta x = L/(m - 1/2)$.

### 2.3 Computational procedure

#### 2.3.1 Finite difference scheme

The governing equations are discretized in space with the finite difference method on the staggered grid of Fig.2 The implicit finite difference form of mass conservation equation over a point $x_{i+1/2}$ with $i=1, 2, \ldots m-1$ with constant time stepping $h$ is:

$$\frac{Q_i^{n+1} - Q_i^n}{h} + gA \frac{H_i^{n+1} - H_i^{n+1/2}}{\Delta x} + \frac{f}{2DA} |Q_i^{n+1/2}| Q_i^{n+1/2} = 0 \tag{7}$$

For the above equation $i=2, 3 \ldots m$.

![Fig.3 Definition of pseudo parameters on upstream nodes](image)

#### 2.3.2 Implementation of Boundary condition

The system has to be completed with appropriate boundary conditions. The boundary conditions are implemented with the following assumption:

Upstream of piping system at nodes $i=1, 2$ the head is not defined, at these points we will define the pseudo-heads $\overline{H}_1$, $\overline{H}_2$ and at node $1 + 1/2$ we will define the pseudo-flowrate $\overline{Q}_{1+1/2}$. With above assumptions we discretize the momentum equation over the node $1 + 1/2$ (Fig.3):

$$\frac{\overline{Q}_{1+1/2}^{n+1} - \overline{Q}_{1+1/2}^n}{h} + gA \frac{\overline{H}_{1+1/2}^{n+1} - \overline{H}_{1+1/2}^{n+1/2}}{\Delta x} + \frac{f}{2DA} |\overline{Q}_{1+1/2}^{n+1/2}| \overline{Q}_{1+1/2}^{n+1/2} = 0 \tag{8}$$

In above equation the pseudo-flowrates are approximated as:

$$\overline{Q}_{1+1/2}^{n+1} = (Q_{1+1/2}^{n+1} + Q_{1+1/2}^n)/2 \tag{9}$$

$$\overline{Q}_{1+1/2}^n = (Q_{1}^{n+1} + Q_{1}^n)/2 \tag{10}$$

And similarly the pseudo-head is approximated as:

$$\overline{H}_{1+1/2}^{n+1} = (H_{1+1/2}^{n+1} + H_{1+1/2}^{n+1/2})/2 \tag{11}$$
At node no. 1 the pseudo-head can be obtained from boundary condition:

\[ H_{n+1}^{i} = H_{res} - \frac{(1+k_{e})}{2gA^2}(Q_{i}^{-1})^2 \]  

(12)

Downstream of piping system at (x=L), for the nodes \( m + 1/2 \) and \( m - 1/2 \) the flowrate is not defined, therefore at these nodes we will define the pseudo-flowrate \( \bar{Q}_{m+1/2} \), \( \bar{Q}_{m-1/2} \) and at node ‘m’ we will define the pseudo-head \( \bar{H}_{m} \) (Fig. 4)

\[ \bar{H}_{m} = \frac{H_{n+1}^{i} + H_{n}^{i}}{2} \]

\[ \bar{Q}_{m+1/2} = \frac{Q_{m+1} + Q_{m}}{2} \]

\[ \bar{Q}_{m-1/2} = \frac{Q_{m} + Q_{m-1}}{2} \]

Fig.4 Definition of pseudo parameters on downstream nodes

Similarly with above assumptions we discretize the mass conservation equation over the node \( m \):

\[ \frac{\bar{H}_{m}^{n+1} - \bar{H}_{m}^{n}}{h} + \frac{a^2}{gA} \frac{\bar{Q}_{m+1/2}^{n+1} - \bar{Q}_{m-1/2}^{n+1}}{Ax} = 0 \]

(13)

And approximate \( \bar{H}_{m}^{n+1} \), \( \bar{H}_{m}^{n} \) and \( \bar{Q}_{m+1/2}^{n+1} \) as:

\[ \bar{H}_{m}^{n+1} = \frac{(H_{m}^{n+1} + H_{m+1/2}^{n})}{2} \]

(14)

\[ \bar{H}_{m}^{n} = \frac{(H_{m+1/2}^{n} + H_{m}^{n+1})}{2} \]

(15)

\[ \bar{Q}_{m+1/2}^{n+1} = \frac{(Q_{m+1/2}^{n} + Q_{m+1}^{n+1})}{2} \]

(16)

At the last node \( m + 1/2 \) the pseudo-flowrate can be obtained from equation (4):

\[ \bar{Q}_{m+1/2}^{n+1} = \frac{Q_{x} \tau(t)}{\sqrt{H_{m+1/2}^{n+1}}} \sqrt{H_{m+1/2}^{n+1}} \]

(17)

### 2.3.3 Initial conditions

The values of the hydraulic parameters at time \( n = 0 \) are given by the initial steady-state conditions. Therefore with the steady state flowrate \( Q_{i}^{0} = Q_{s} \) the initial profile of piezometric head along the pipe can be calculated as:

\[ H_{x}^{0} = H_{res} - \frac{(1+k_{e})}{2gA^2} \frac{Q_{s}^{2}}{D} \]

(18)

### Table 1 Parameters for valve closure Experiment

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>pipe material</td>
<td>commercial steel</td>
</tr>
<tr>
<td>Wave Speed</td>
<td>1125 m/s</td>
</tr>
<tr>
<td>internal diameter</td>
<td>0.101 m</td>
</tr>
<tr>
<td>Initial Discharge</td>
<td>0.0165 m³/s</td>
</tr>
<tr>
<td>Upstream Tank Head</td>
<td>2.65 m</td>
</tr>
<tr>
<td>Entrance Loss Coefficient</td>
<td>1.44</td>
</tr>
<tr>
<td>Friction Factor</td>
<td>0.0223</td>
</tr>
<tr>
<td>Initial Valve Position</td>
<td>( \tau = 0.62 )</td>
</tr>
<tr>
<td>Density</td>
<td>999.1 Kg/m³</td>
</tr>
</tbody>
</table>

### 3 Verification of numerical model

The system equations have been solved by full implicit FD method. The model is verified by using experimental data obtained by Silva Araya [7] in a single pipe system. The arrangement consists of a constant-head reservoir open to the atmosphere connected by a horizontal steel pipe to a motorized butterfly valve located 32 m downstream of the reservoir.

Recorded transient heads at valve upstream point and pipe midpoint were selected for comparison.

The transient conditions for this case were produced by a gradual closure of the downstream valve. The relative valve opening (\( \tau \) verses time (t) curve is specified in a tabular form by Silva-Araya [7].

The pressure surge into the piping system is highly sensitive to the valve characteristic curve especially near the end of the closure time. Therefore to compute the transient-state conditions for closing valve, here cubic-spline interpolation was used to compute the effective valve opening (\( \tau \)) from the tabulated data during the simulation of the valve operation. Other data used for simulation are listed in Table 1.

Fig.5 and Fig.6 compare the computed pressure heads in both proposed computation method and
method of characteristic. These comparisons are done at the valve upstream and at the L/2 of the pipe.

While comparing the graphs, one can notice that the maximum pressure head and oscillation frequencies are predicted uniformly by both methods. However, the predicted damping of the pressure wave amplitude in proposed method is more intensive compared with the calculated one by method of characteristic.

The results of the proposed computation method and experimental data are shown in Fig.7 and Fig.8.

Again the previous points in system are considered for comparison. It is clearly seen that calculation using the proposed method is much closer to the experimental data and almost all pressure head picks are in the same range of amplitude for both valve upstream position and at the midpoint of the pipe. Additionally, the rate of pressure damping is in close agreement with measured data by Silva-Araya.

However, a shift in the frequency of pressure oscillations was present between the computed and experimental data for both positions. The mention
shift is probably a consequence of the measuring equipment.

4 Summary and conclusions
Implicit Finite Difference scheme using non-symmetrical staggered grid is proposed for solving water hammer equations. The results of proposed method are compared with experimental data and the results of solving method of characteristic. The obtained results point out that it is possible to predict pressure fluctuations (amplitude and rate of damping) accurately in the water hammer with proposed method.

The good agreement between the results of computation method and experimental data can be due to accuracies with considering second order spatial discretization and accurate implementation of boundary conditions in the proposed staggered grid.

Acknowledgment:
The writers wish to thank the Research and Development Center of Pars Oil and Gas Company for the support to this work.

References: