International Journal of Fluid Power 3 (2002) No.2 pp. 17-27

A FINITE VOLUME CENTRAL DIFFERENCING SCHEME FOR SIMULATION OF THE SHUT DOWN PROCEDURE OF A HYDRAULIC SYSTEM

Karim Mazaheri and Reza H. Farahani

Sharif University of Technology, Tehran, Iran 11365-9567
mazaheri@sharif.edu

Abstract

A new central differencing finite volume scheme is investigated for solution of unsteady hydraulic problems as water hammer in pipe systems. Special time stepping procedure similar to Runge-Kutta algorithm is used to stabilize this second order scheme. It is monotonized by adding dissipative terms including second and fourth derivatives of the conserved variables, with coefficients proportional to derivatives of pressure or volumetric flow, which keeps the second order of accuracy in smooth flow regions. The one-dimensional unsteady incompressible equations are solved for a water hammer situation, and results are compared to existing analytical solutions. Results are also compared with numerical results of classical characteristic method, which is proved to be fairly accurate. The scheme could easily be generalized to two-dimensional case. Finally this procedure is used for analysis of the shut down procedure of a hydraulic system. Components of the system are modeled and effects of important parameters on the performance are studied.

Keywords: water hammer, hydraulic system analysis, finite volume schemes, shut down procedure

1 Introduction

The very high-pressure pulses or shocks caused by rapid flow rate change in a pipe or a system of pipes (also called *water hammer*) could result in catastrophic breaking of the hydraulic transmission lines and may also destruct equipment and components installed on the line. The rapid change is a result of performance of pumps, turbines, and valves in the hydraulic system. Water hammer can reach pressure levels far exceeding the over pressure rating of our components. This phenomenon is mostly a function of six parameters: length, diameter, velocity, the rapid change characteristic time, and the fluid density and sound speed. Of course most other system components and properties will also have contribution to this phenomenon. To protect a system from water hammer damage, three systems are usually used to eliminate or control the pressure rise. These are surge tanks, slowing operation and pressure snubbers. To estimate the amount of this pressure rise and effect of different devices in control of it, we may use numerical simulation of the flow field and components.

Many researchers have simulated this unsteady phenomenon. During 1920-1950 graphical methods were extensively used to solve the *water hammer* equations. In these methods, convective and friction terms are neglected, which may be quite effective. Also their application for a system of pipes is difficult. Streeter (1988) applied the characteristics method to the water hammer equations. Many people since then have used characteristics method with different sets of assumptions. Lai (1988) and Katopodes and Sterlkoff (1978) also used this method for their analysis.

During 1980s many people used finite difference and finite volume formulations to simulate this flow pattern. The main advantages of these methods in the respect to classic method of characteristics (MOC) are higher accuracy, ease of usage especially if one needs to increase the spatial dimensions, ease of boundary condition application and easier simulation of other hydraulic components, although usually the computational speed of the MOC is higher. Some of these methods are: Garcia-Navaro (1986), Fennema and Chaudhry (1987), Rao and Latha (1992) and Nujic (1995) (using Lax-Friedrich's scheme), and Savic and Holly (1993) (using Godunov scheme). Leaf and Chawala (1979) also presented the finite element solution of this problem. They showed that, other than simpler boundary conditions applications, this formulation has no advantage to other schemes. Recently *shock-capturing* finite difference schemes, which are originally applied to the gas dynamic Euler equations, are also used. These schemes apply an upwind difference for-

This manuscript was received on 7 February 2001 and was accepted after revision for publication on 14 June 2002

mulation to the linearized Riemann problem. Finite volume formulation has certain advantages in simulation of physical phenomena including discontinuities. Also, they could be easily applied to structured and unstructured grids and do not require coordinate transformation. In hydraulic applications, it was first used to solve unsteady free surface problems. Bellos and Sakkas (1991), Zhao (1994), Garcia-Navaro et al (1994) and Mingham and Causon (1998) have presented some results in this regard.

In gas dynamics community, these schemes have a longer history of usage, which returns to 1970s. Some of the most important results in this field are introduced by Roe (1981) and Jameson and Turkel (1981), who used upwind and central differencing schemes, respectively. One of the main advantages of the finite volume schemes is that one virtually can incorporate any parts of the physics of the flow field that he wishes (like unsteady viscous momentum and heat transfer effects), but of course he should pay for this ease of usage, since he solves the equations in the physical space.

In this paper, we have developed this central differencing finite volume scheme for the hydraulic equations, especially for simulation of unsteady flow in a system of pipes. Most high order accurate FVM schemes suffer from non-monotonicity near discontinuities like a pressure jump. This scheme also keeps the monotonicity of pressure distribution near the pressure jump. For this reason, this scheme is well suited for discontinuities simulation, like a very rapid pressure rise in a water hammer. The scheme is second order accurate, except in the vicinity of discontinuities, where it is of first order of accuracy. To stabilize the scheme, based on Fourier analysis, a special multi-staging procedure is used which is very similar to the Runge-Kutta time integration. The second and fourth order derivatives of conserved variables with coefficients proportional to the second order derivatives of pressure and flow rate are used as dissipative terms to make the solution near the discontinuities monotone. These will make this second order scheme only first order accurate in the non-smooth regions.

Here, the governing equations are first introduced. Then fundamentals of central differencing finite volume schemes and how they are applied to the hydraulic equations are presented, and after explaining how initial and boundary conditions are applied, and how we march through the time, two applications are shown. The first application is the solution of the hydraulic equations in a simple pipe, for which the exact solution is known, and the numerical results are compared with the known results. The second application is a pipe with a control valve at one end, where its effect on the flow field in different situations will be studied and compared with other verified numerical schemes. At the end a hydraulic system, including a damping sphere with an on-off valve at one end, is simulated, and a parametric study on performance of the system is done, which has provide some valuable information for the system designer.

2 Governing Equations

The governing equations of the unsteady incompressible flow inside a variable area pipe, i.e. the continuity and momentum equations, in the one-dimensional case are:

$$\frac{\partial U}{\partial t}+\frac{\partial F}{\partial x}+S=0 \tag{1}$$

$$U=\begin{bmatrix} h \\ Q \end{bmatrix} \qquad F=\begin{bmatrix} \frac{a_L^{\,2}}{gA}\;Q \\ gA\;\; h \end{bmatrix} \qquad S=\begin{bmatrix} 0 \\ \frac{fQ|Q|}{2AD} \end{bmatrix} \tag{2}$$

where U is a vector, which in comparison to gas dynamics equations is called the conservative variables vector, and F is also a vector, which is similarly called the flux vector. A is the cross sectional area, Q is the volumetric flow, g is the gravitational acceleration, h is the flow head, f is the friction factor, D is the diameter and a_L is the sound speed in the liquid. In this approximation (Streeter, 1988), incompressibility means that density is assumed constant, while we allow to the sound speed in the fluid to be finite.

3 Central Difference Finite Volume Formulation

During the last two decades, Euler and Navier-Stokes equations have been the most popular equations for aerodynamic and gas dynamic analysis. Although the Euler equations use the inviscid flow assumption, still one may capture numerically the shock waves and respectively, most other flow properties correctly. Therefore, many different methods have been suggested for solution of these equations. The original finite difference methods suggested for solution of these equations were either inaccurate or suffered from lots of computations.

In 1980's two new families of finite volume methods were developed, both of which are still in common use. The first family, which is already applied to the hydraulics equations, are *the upwind schemes*. The second one is *the central differencing scheme*, and in this paper it is applied to the hydraulic equations. Generally, in finite volume formulation, one integrates the governing equations on each individual cell of the flow field. Based on our governing equation, our cells are cylindrical, with only one independent space variable, x. Using the Gauss theorem, the volume integral of the flux term is changed to a surface integral:

$$\frac{\partial}{\partial t}\iiint_{\Omega} U d\Omega+\iint_{\partial\Omega} F ds+\iiint_{\Omega} S d\Omega=0 \tag{3}$$

where Ω is the control volume with boundary surface $\partial\Omega$, $d\Omega$ is the control volume differential element, with differential surface ds. Application of this equation to each cell (semi-discretization) produces an ordinary differential equation, which could be written in this form:

$$\frac{d}{dt}(V_i U_i) + \sum_{\substack{\text{faces} \\ \text{of cell } i}} (Fs) + S_i = 0 \tag{4}$$

where V_i is the volume of the i^{th} cell, which is not a function of time. $\sum_{\text{faces}} (Fs)$ is the net flux exiting the i^{th} control volume, and in our one dimensional approximation is equal to

$$\sum_{\substack{\text{faces} \\ \text{of cell } i}} (Fs) = (Fs)_{i+\frac{1}{2}} - (Fs)_{i-\frac{1}{2}} \tag{5}$$

$$(Fs)_{i+\frac{1}{2}} = [(Fs)_i + (Fs)_{i+1}]/2 \tag{6}$$

Value of F at i+1/2 is the average of values of F in cells i and i+1. This will assure second order of accuracy, of course if the grid shape is smooth enough. It is well known that this central differencing scheme is unstable, and that because of this second order of accuracy, the solution is not smooth enough close to the flow discontinuities. Many tools are developed to remove these deficiencies. One of them is using second and fourth order derivatives of flow variables as dissipative terms, which are added as ($d_{i+1/2}$ - $d_{i-1/2}$) to the left hand side of Eq. 4, where:

$$d_{i+\frac{1}{2}} = \frac{V_{i+\frac{1}{2}}}{\Delta t} \left(\varepsilon^{(2)}_{i+\frac{1}{2}}(U_{i+1} - U_i) - \varepsilon^{(4)}_{i+\frac{1}{2}}(U_{i+2} - 3U_{i+1} + 3U_i - U_{i-1})\right) \tag{7}$$

The second order dissipative terms added to the governing equations in the neighborhood of flow discontinuities will remove all oscillations and under and overshoots. These terms have coefficients proportional to some flow gradient, which are of second order of magnitude in smooth flow regions and are of first order of magnitude in the non-smooth regions. One may define these coefficients using second order difference formula for the conservative variable U, or some other relevant parameter like pressure:

$$\upsilon_i = \frac{|p_{i+1} - 2p_i + p_{i-1}|}{|p_{i+1}| + 2|p_i| + |p_{i-1}|} \tag{8}$$

Now one defines $\varepsilon^2_{i+\frac{1}{2}}$ and $\varepsilon^4_{i+\frac{1}{2}}$ as:

$$\varepsilon^2_{i+\frac{1}{2}} = \min\left\{\frac{1}{2}, k^{(2)} \max(\upsilon_i, \upsilon_{i+1})\right\},$$
$$\varepsilon^4_{i+\frac{1}{2}} = \max(0, k^{(4)} - \varepsilon^2_{i+\frac{1}{2}}) \tag{9}$$

where typical values for $k^{(2)}$ and $k^{(4)}$are respectively 1/4 and 1/256. The fourth order dissipative terms should be turned off close to the discontinuities, which is achieved by the above definitions. In the smooth regions the accuracy of solution is second order, and the dissipative terms are of higher order, but close to the discontinuities one will lose high order of accuracy to gain monotonicity of the solution.

4 The Multi-Stage Time-Stepping And Numerical Stability

Since the procedure used here is explicit, *the Courant number* $\upsilon = a_L \Delta t/\Delta x$ is controlled to assure the stability. Here a_L is the sound speed, Δx is the length of each cell ($\Delta x = L/(a_L N)$), where L is the length of the flow field and N is the number of cells. The central differencing scheme is unstable for all values of υ. One may show that using a special time integration procedure could stabilize this scheme (Jameson, 1981). This procedure is called multi-staging. To see this, one may look at a model partial differential equation like:

$$\frac{\partial U}{\partial t} + a\frac{\partial U}{\partial x} + \varepsilon \Delta x \frac{\partial^2 U}{\partial^2 x} = 0 \tag{10}$$

defined on a uniform grid with spacing Δx. If ∂_x and ∂_{xx} are discretized, one finds:

$$\frac{dU_i}{dt} + \frac{a}{\Delta x}(U_{i+1} - U_{i-1}) + \frac{\varepsilon}{\Delta x}(U_{i+1} - 2U_i + U_{i-1}) = 0 \tag{11}$$

Using Fourier transform, one finds a simple ODE:

$$\frac{d\hat{U}}{dt} + \lambda \hat{U} = 0 \tag{12}$$

where λ is a complex number (Jameson, 1981). Maximum allowable value of the imaginary part of $\lambda\Delta t$ determines the stability margin. The time integration procedure also affects the Fourier footprints, and the stability margin. One form of this procedure is the four-stage Runge-Kutta integration procedure:

$$\begin{aligned} U^0 &= U^n \\ U^1 &= U^0 - \alpha_1 (res)^0 \\ U^2 &= U^0 - \alpha_2 (res)^1 \\ U^3 &= U^0 - \alpha_3 (res)^2 \\ U^4 &= U^0 - \alpha_4 (res)^3 \\ U^{n+1} &= U^4 \end{aligned} \tag{13}$$

Here, (*res*) is the residual of computation, which is equal to $\partial U/\partial t$. Different values of α_i will provide different properties for convergence. The standard Runge-Kutta has the following coefficients:

$$\alpha_1 = \frac{1}{4} \qquad \alpha_2 = \frac{1}{3} \qquad \alpha_3 = \frac{1}{2} \qquad \alpha_4 = 1 \tag{14}$$

The Courant number corresponding to these coefficients is $2\sqrt{2}$. Addition of dissipative terms will not deteriorate this stability margin.

Initial and Boundary Conditions

Initial and boundary conditions are very dependent on the system components. For a simple pipe, the energy equation is used to find the initial pressure and flow. The initial flow Q_0, and the pressure loss R, is uniform through the pipe:

$$Q_0 = \sqrt{\frac{h_{\mathrm{res}}}{\frac{f\,L}{2gDA^2} + \frac{1}{2gC_{\mathrm{V}}{}^2}}} \qquad (15)$$

$$R = \frac{h_{\mathrm{res}} - h_0}{NQ_0^2} \qquad (16)$$

$$h_0 = \frac{Q_0^2}{2gC_{\mathrm{V}}{}^2} \qquad (17)$$

where h_{res} is the upstream reservoir head, C_{V} is the discharge coefficient, f is the pipe friction factor, L is the pipe length and D is the pipe diameter (Streeter, 1981).

For boundary conditions, we use standard characteristics method, which results in the following relations (Streeter, 1981):

$$h = C_{\mathrm{P}} - BQ_{\mathrm{P}} \qquad (18)$$

$$Q_{\mathrm{P}} = -gBC_{\mathrm{V}}^2 + \sqrt{((gBC_{\mathrm{V}}^2)^2 + 2gC_{\mathrm{V}}^2C_{\mathrm{P}})} \qquad (19)$$

$$B = \frac{a_{\mathrm{L}}}{gA} \qquad (20)$$

5 Results

To verify accuracy of the procedure introduced for solution of hydraulic problems, i.e. simulation of pressure waves in system of pipes, we analyze a simple pipe using this procedure and compare our results with results of the classical method of characteristics (MOC). Also we will add a valve to the downstream end of this system and will study effects of closing and opening the valve on the flow field. At the end, we will apply our scheme to a real engineering problem, and will accomplish a parameter study, which is very useful in the design procedure.

5.1 Rapid shut down procedure in a simple pipe

Consider a simple pipe placed between a reservoir and an on-off valve, which is initially closed, and is opened at time $t = 0$. The length to diameter ratio is 10. Of course the one-dimensional approximation is more accurate as the ratio of length to diameter increases. A ratio of 10 is really very small, but still it is acceptable (One of the advantages of FVM is that you could easily generalize it to two or three-dimensional approximation, then there will be no lower limit for this ratio. Here it does not worth to do it.). We have used 80 similar cylindrical cells to solve this problem. The sound speed is 1200 m/s. One uses the maximum stable Courant number for this calculation, which corresponds to time step equal to 0.01 sec. Friction factor is assumed to be 0.022. Figure 1 shows results for pressure variations at the downstream end of the pipe. The central differencing and characteristic methods solutions are compared. Figure 2 compares the same methods for the velocity variations. Note that the characteristic method produces almost the exact solution. Results agree very well, which shows the consistency and stability of the FVM scheme.

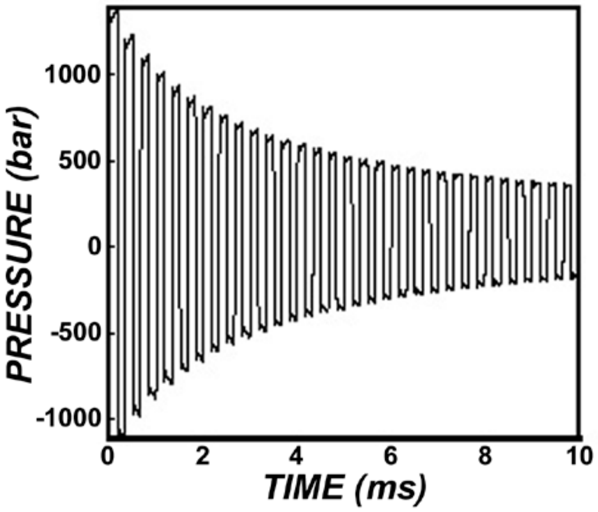


Fig. 1a: *Pressure versus time at the end of the pipe, central differencing method*

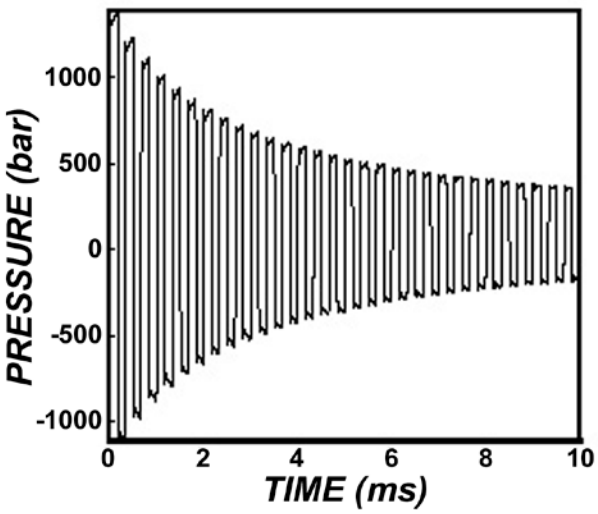


Fig. 1b: *Pressure versus time at the end of the pipe, characteristic method*

We may use our simulation for a parameter study to see the effect of each parameter on performance of the pipe system. Figure 3 shows effect of five-times increase of the pipe length, every other thing is kept constant. Again pressure variations for both methods of FVM (central differencing) and characteristics method are compared and agree with each other very well. As expected, the most noticeable change is decrease of the frequency of variations by a factor of five, other parameters are not changed significantly. Similarly, Fig. 4 shows the same effects for diameter increase. As we expect, the diameter effect on the performance and the wave pattern is very small, for obvious reasons.

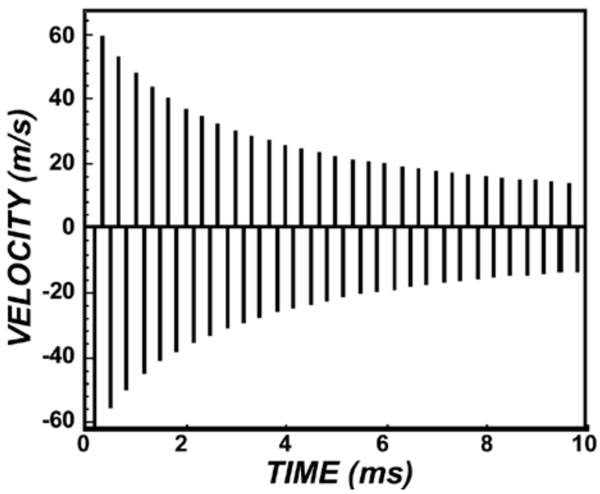


Fig. 2a: *Velocity versus time at the end of the pipe, central differencing method*

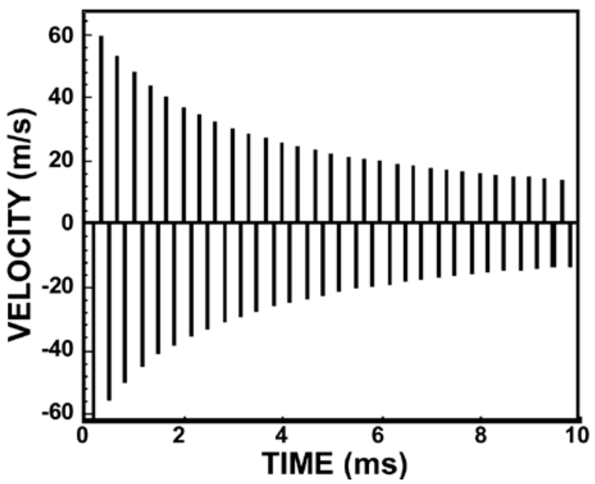


Fig. 2b: *Velocity versus time at the end of the pipe, characteristic method*

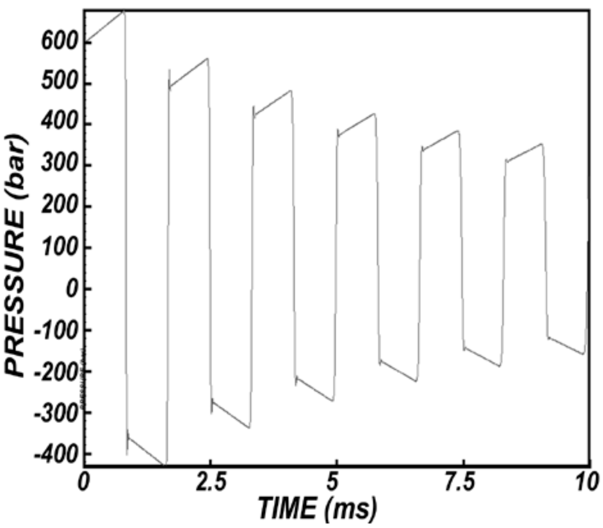


Fig. 3a: *Pipe length increase effect, FVM*

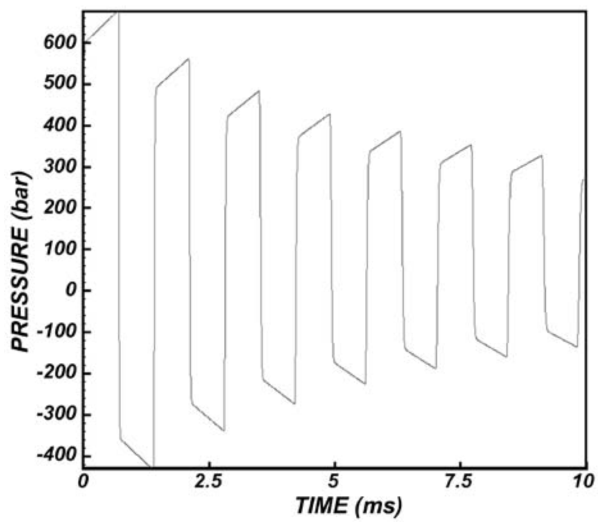


Fig. 3b: *Pipe length increase effect, characteristic method*

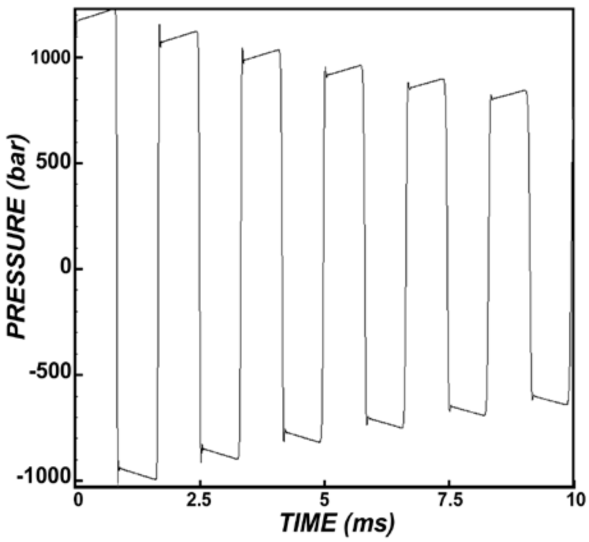


Fig. 4a: *Pipe diameter increase effect, FVM*

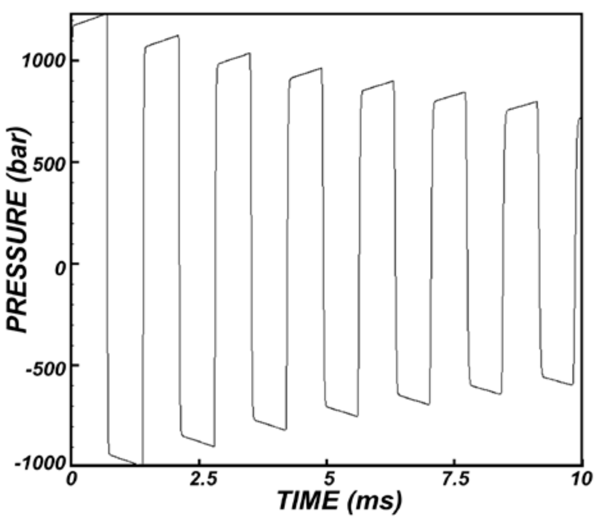


Fig. 4b: *Pipe diameter increase effect, characteristic method*

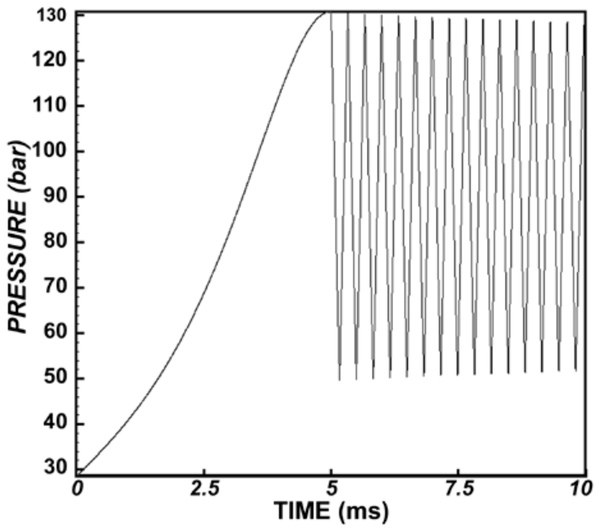


Fig. 5a: *Valve closing procedure for T=0.005 s, FVM*

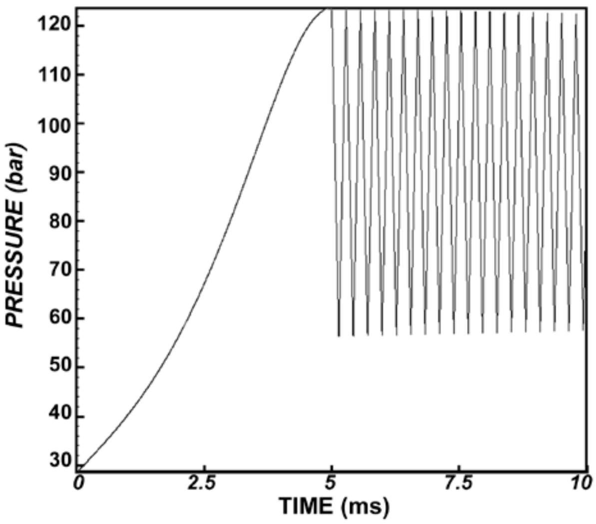


Fig. 5b: *Valve closing procedure for T=0.005 s, characteristic method*

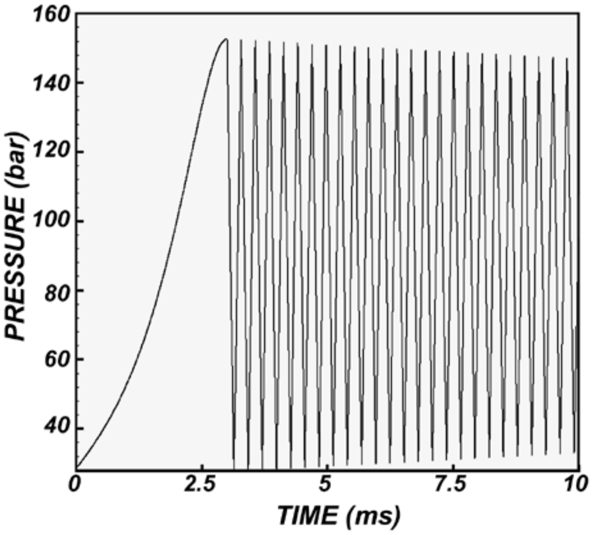


Fig.6a: *Valve closing procedure for T=0.003 s, FVM*

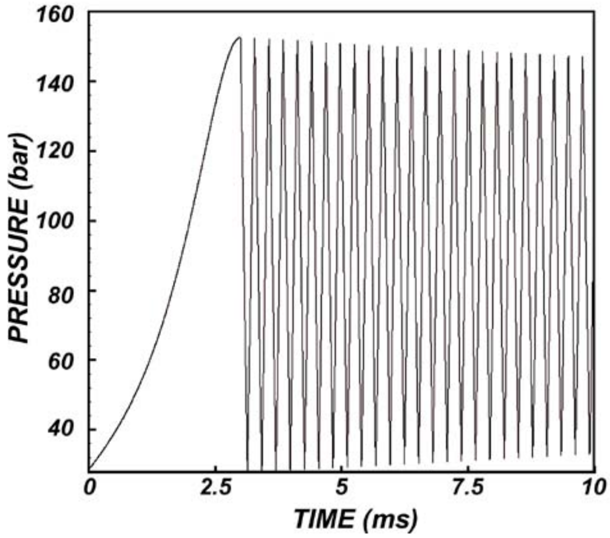


Fig.6b: *Valve closing procedure for T=0.003 s, characteristic method*

Now we simulate closing of a valve at the end of a pipe. We assume the valve closing procedure has a linear form, i.e. $C_V = (1-t/T)A$, where T is the total time of closing the valve. The valve is completely open at $t = 0$. Again pressure variations at the downstream end of the pipe are computed by both FVM and characteristic methods. Figures 5 and 6 show these results for respectively T = 0.005 and 0.003 seconds. As one may observe, the amplitude and frequency is simulated very well. We have done a grid size study, which shows if we increase the number of cells, we could get even better agreement between them.

5.2 A Hydraulic System

The hydraulic system studied here is shown schematically in Fig. 7a. This system includes four main components: a pipe, a damping sphere, a connection pipe and a closing valve. The system is used as part of another fairly complex hydraulic system, and should terminate the liquid flow in a fraction of time. Connection of the sphere and the connection pipe is realized via a diaphragm, which separates fluid and air, and the sphere is initially filled by air. At the start, the valve is open and flow goes through the pipe to the exit point. At time $t = 0$ the valve is closed pyrotechnically by receiving an electrical signal and the diaphragm is broken, which allows liquid to enter to the sphere, and absorb part of the pressure pulse. The total closing time T is 0.003 seconds. We would like to investigate the effect of different parameters on the performance of the system, especially on the pressure history, since pressure overshoots may severely damage the flow line. The closing valve is an on-off type, and we assume the valve closing procedure has a linear form. To derive the damping sphere dynamic equation, one may start with the continuity equation, with the perfect gas assumption:

$$\frac{dM}{dt} = \frac{V}{RT}\frac{dp}{dt} \qquad (21)$$

where M is mass of the gas in the sphere. For an isentropic process, since we have p/p^{γ}=const. or $\rho=\rho_0(p/p_0)^{1/\gamma}$, therefore:

$$\frac{dM}{dt} = \frac{V\rho_g}{\gamma p}\frac{dp}{dt} \tag{22}$$

At the same time, since the liquid mass in the sphere is $M_L = \rho_L V_L$, we have

$$\frac{dM_L}{dt} = V\frac{d\rho_L}{dt} + \rho_L\frac{dV_L}{dt} \tag{23}$$

Since variations of liquid volume are negligible, the second term in this equation is ignored. Using the definition of the bulk modulus of compressibility, the differentiation of liquid density is replaced by the pressure variations:

$$\frac{dM_L}{dt} = \frac{V_L}{{a_L}^2}\frac{dp_L}{dt} \tag{24}$$

where a_L is the speed of sound wave in the liquid. Combination of Eq. 22 with 24 results in:

$$\left(\frac{V_L}{{a_L}^2} + \frac{V\rho_g}{\gamma p}\right)\frac{dp}{dt} = \frac{d(M_L + M)}{dt} \tag{25}$$

Note that the right hand side is the mass flow into the sphere, so finally we will have:

$$\frac{dp}{dt} = \dot{m}\left(\frac{V_o - V}{{a_L}^2} + \frac{\rho_g V}{\gamma p}\right)^{-1} \tag{26}$$

where p is the pressure inside the sphere, V_0 is the sphere volume, V is volume of the gas contained in the sphere, with density ρ_g, specific heat ratio γ. The fluid mass flow into the sphere is:

$$\dot{m} = \mu\, A_D\sqrt{2g(p_p - p)}\; Sign(p_p - p) \tag{27}$$

where p_p is the upstream pressure of the inlet diaphragm of the sphere, μ is the mass flow coefficient (around 0.6), and A_D is the inlet area. The boundary condition upstream of the main pipeline is assumed to be a constant pressure (head) reservoir, similar to the previous examples.

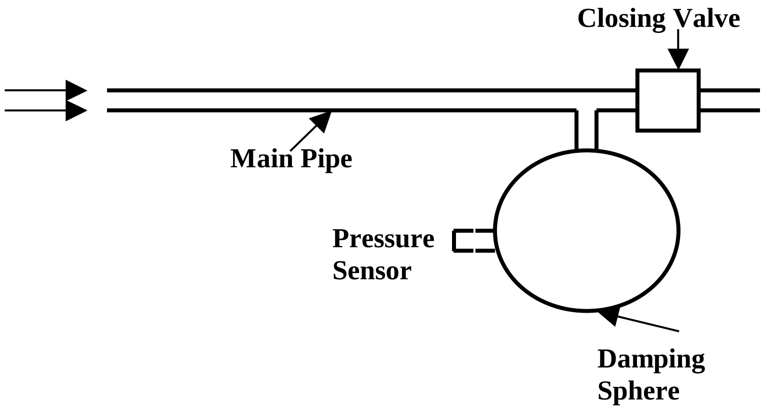


Fig. 7a: *Schematic of the hydraulic system*

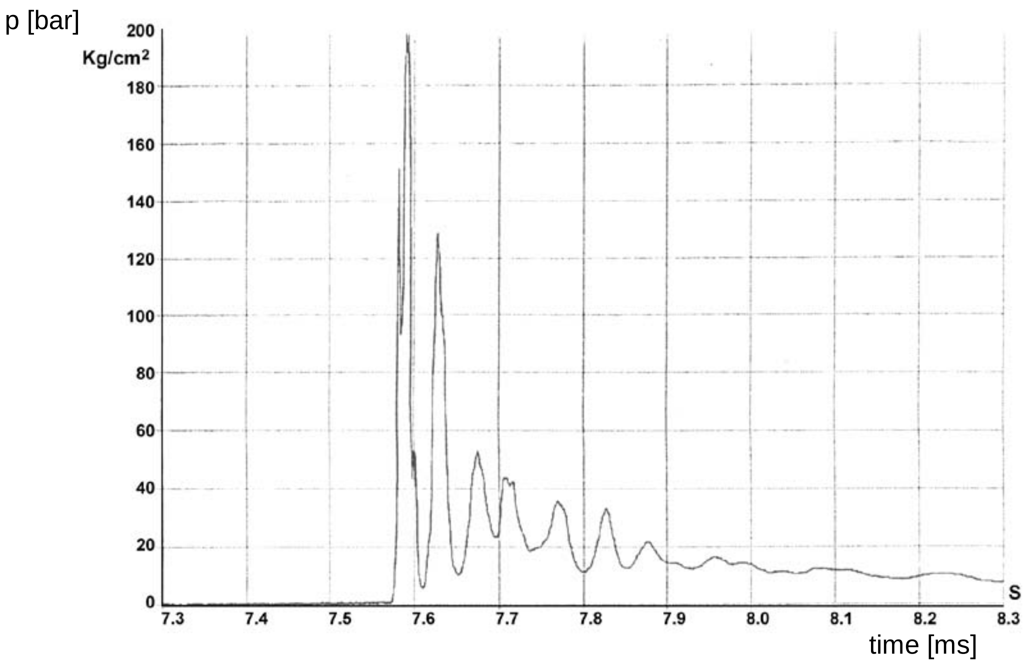


Fig. 7b: *Measured pressure variation in the damping sphere*

To verify accuracy of the simulation of this system, experimental results showing pressure variations for typical shut down procedure is used. The experimental results are achieved in the test of another fairly complex turbopump hydraulic system, which includes the system shown in Fig. 7a in its pumps downstream path. The whole system is not simulated here. The fluid in the line is nitric acid, with specific gravity equal to 1.55, viscosity equal to 1.45 (centipoise), and temperature equal to 297 K. Length and diameter of the main pipe are respectively 362.5 and 48 mm, the connection pipe length and diameter are respectively 55 and 32 mm, and the radius of the damping sphere is 65 mm. The pipe friction factor is assumed to be 0.022. The stagnation pressure of the pump is 90 bar, and the mass flow is 77 kg/s. In the shut down process, the turbine gas generator is instantly turned down, but the high speed and inertia of the turbopump, tends to keep the pressure rise very high. At the same time, the valves in the downstream path are closed, the high pressure generated in this way breaks the inlet diaphragm of the damping sphere (at about 150 bar), and the damping sphere and connection pipe act as a damper to absorb part of the water hammer effect on the pressure rise. Upstream of the pump, a constant pressure reservoir is used. Other details of system are believed to have negligible effects on the dynamic performance. The measurment was achieved by medium frequency pressure sensors, the data acquisition system used standard high speed I/O cards with a minimum sampling rate of 400 Hz for each channel.

Pressure in the damping sphere (Fig. 7b) and many other locations were measured. These experimental results show an overshoot of about 200 bar, oscillating with a frequency of 20 Hz, and damping to 50 % of the initial amplitude in 0.3 seconds. Results are compared with this simulation of the system. To avoid effects of system components on our analysis, their effects were considered in the boundary conditions. The simulation was able to predict the overshoot pressure very well, but the damping coefficient was over predicted. This is mostly because we have not simulated precisely the upstream of this system, which results in unaccurate boundary conditions. Still it is acceptable as a comparative tool to show effects of different parameters on the system performance.

A set of parametric studies were performed to observe and measure effects of the different parameters on the performance. Summary of these studies follows:

(1) Radius of the damping sphere: For R = 0.068, 0.088 and 0.1 m pressure variations are computed. These results are shown in Fig. 8. As one sees, about 30 % increase in the diameter, which is equivalent to 100 % increase of the volume of the damping sphere, may decrease the pressure peak by 20 bar.

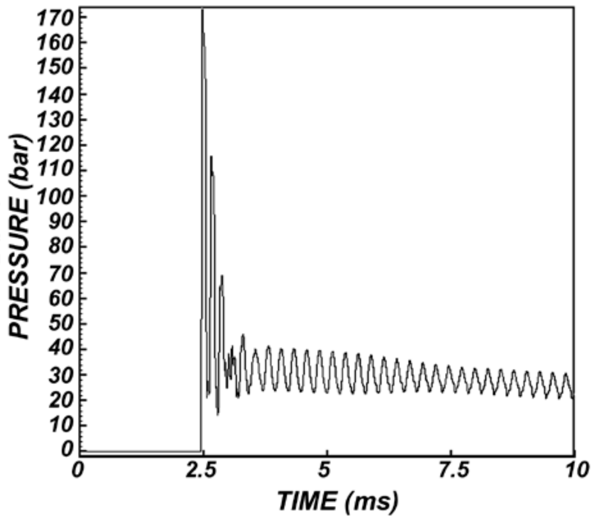


Fig. 8a: *Pressure (P) versus time in damper for R=0.068 m*

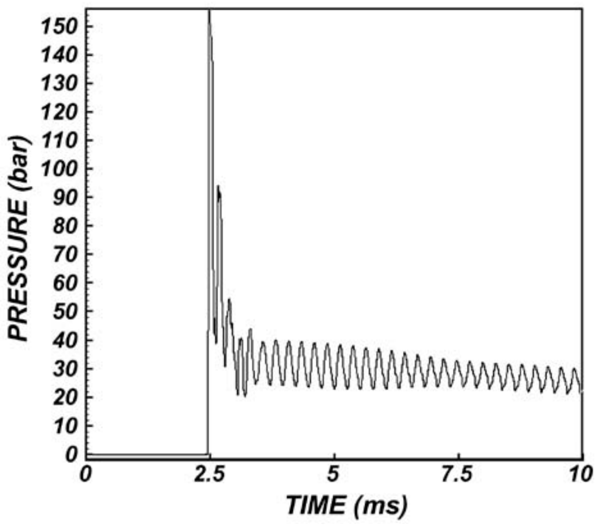


Fig. 8b: *Pressure (P) versus time in damper for R=0.088 m*

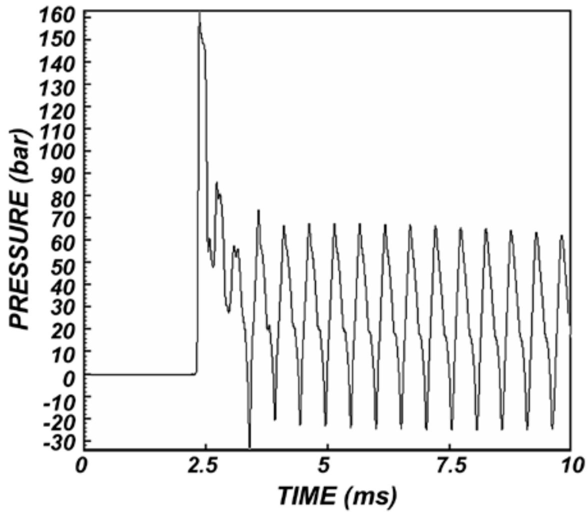


Fig. 8c: *Pressure (p) versus time in damper for R=0.1 m*

(2) Diameter of the connection pipe: For only 20 % increase in the connection pipe diameter, variations in p are not sensible, but the effect on p_p is considerable and is shown in Fig. 9. This means that the connection pipe diameter only affects very weakly the discharge to the damping sphere, therefore, its diameter is mostly effective on the pressure history inside itself.

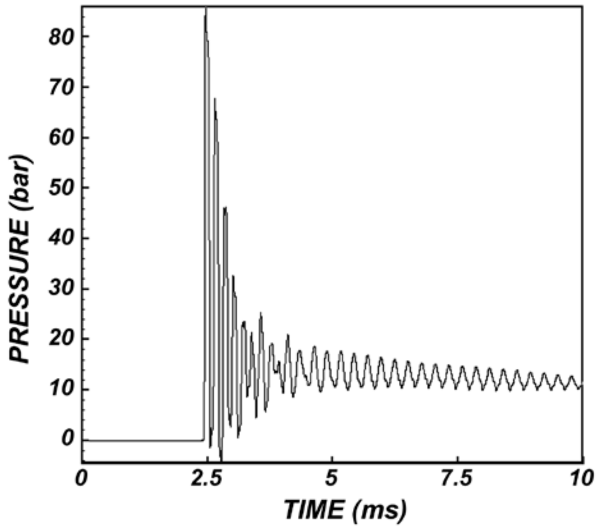


Fig. 9: *Pressure (p) versus time in damper for 20% increase of damper diameter*

(3) Total closing time of the valve (*T*): For T = 0.001, 0.003 and 0.006 s, history of p and p_p are computed and result are shown in Fig. 10a, b, c and 11a, b, c. As one expects, this parameter is the most effective one. An increase of 0.002 seconds has decreased the pressure peak from 200 bar to 175 bar, and next increase by 0.003 seconds, again has decreased it to 145 bar. This change also decreases the pressures variation amplitudes as well, which is generally desirable.

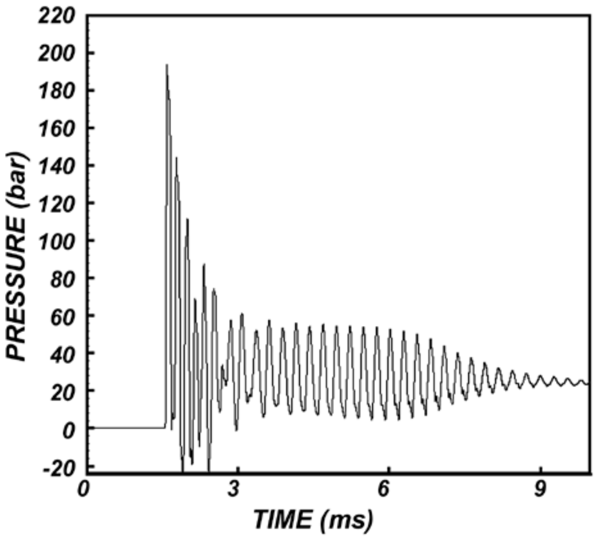


Fig. 10a: *Pressure (p) versus time in damper for total valve closing time of T = 0.001 s*

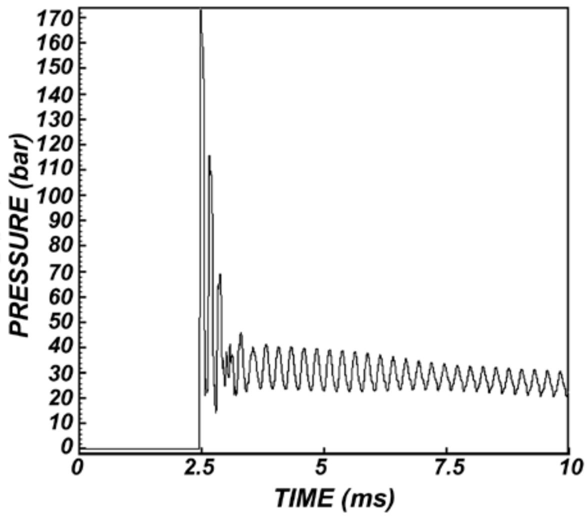


Fig. 10b: *Pressure (p) versus time in damper for total valve closing time of T = 0.003 s*

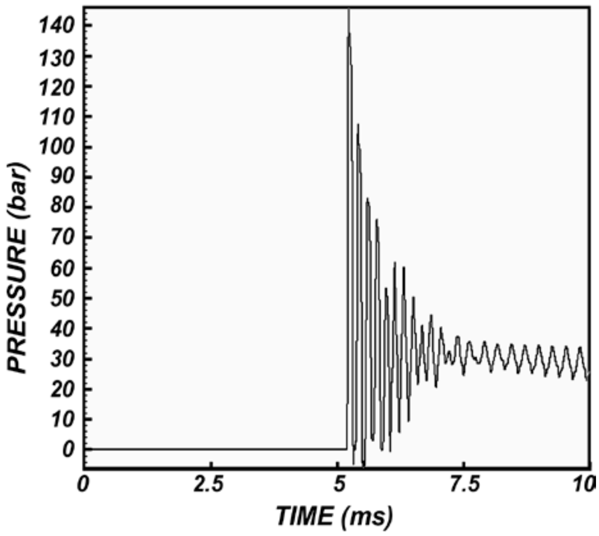


Fig. 10c: *Pressure (p) versus time in damper for total valve closing time of T = 0.006 s*

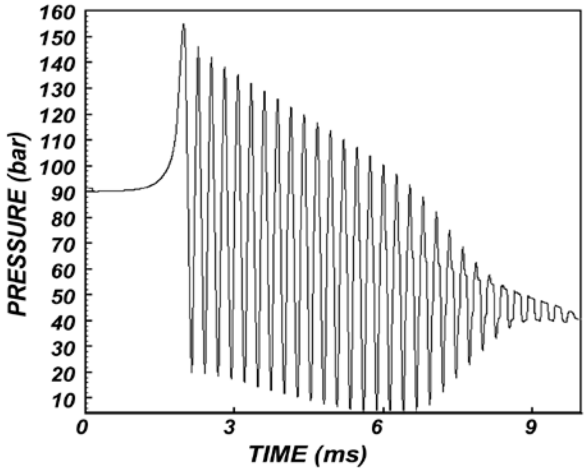


Fig. 11a: *Pressure variations (p_p) versus time at the end of the pipe for valve closing time of T = 0.001 s*

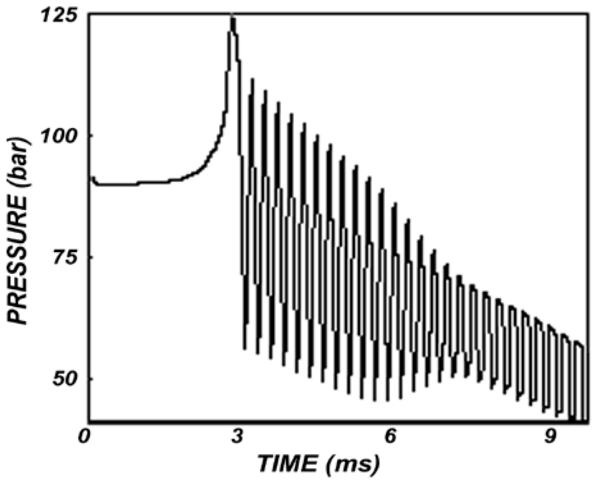


Fig. 11b: *Pressure variations (p_p) versus time at the end of the pipe for valve closing time of T = 0.003 s*

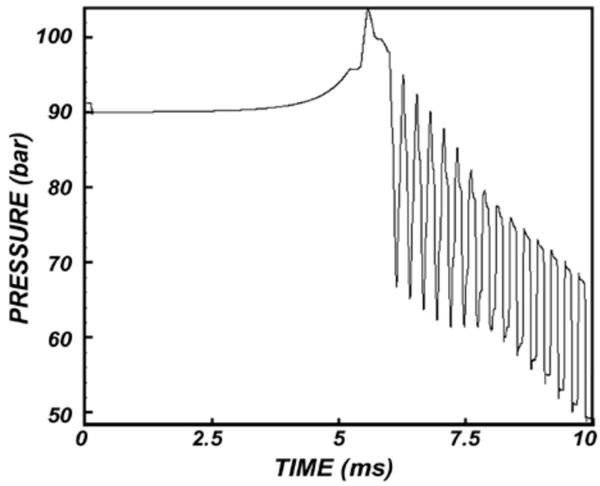


Fig. 11c: *Pressure variations (p_p) versus time at the end of the pipe for valve closing time of T = 0.006 s*

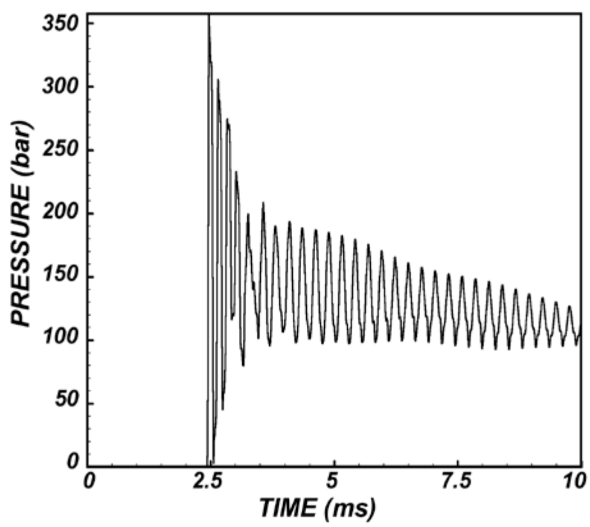


Fig. 12: *Pressure variations (p_p) by increase sphere inlet area by 500%*

(4) Sphere inlet area: Its increase by 500% was studied and results are shown in Fig. 12. This figure shows that this variation makes the sphere less effective in absorbing the pressure pulse, and decreases its surging capacity.

The following results were obtained:

- Only partial increase in the volume of the damping sphere is effective in the system performance.
- Diameter of the connection pipe is fairly ineffective on the performance.
- Increase of *T* considerably decreases the overshoot pressure.
- Increase of inlet area of the damping sphere, increases the undesirable pressure peak.

6 Summary and Conclusions

A new central differencing finite volume formulation for numerical solution of the hydraulic equations is presented. This scheme has been in use for solution of the Euler equations for many years. Advantages of this method to the other finite volume schemes are its speed and accuracy, especially in multi-dimensional problems. The scheme is formulated and is applied to solve a few simple problems to show its applicability. Results are also compared to known results to show its accuracy. The method is also used for analysis of a system of pipes and valves in a rather complex hydraulic system. A parametric study is done, which helps in optimization of the system design. Extending this analysis method to compound pipeline systems, and to more complex hydraulic systems will be investigated in the next steps.

Nomenclature

A	Cross-sectional area of the pipe	[m^2]
a_L	Sound wave speed in the liquid	[m/s]
B	Pipe constant a_L/gA	[s/m^2]
C_p	C$^+$ characteristic value	[m]
C_V	Discharge coefficient	[m^2]
D	Diameter	[m]
$d_{i+1/2}$	Numerical derivative operator	[-]
f	Darcy-Weisbach friction factor	[-]
g	Acceleration due to gravity	[m/s^2]
h	Piezometric head	[m]
h_0	Initial piezometric Head	[m]
h_{res}	Head of uniform flow	[m]
i	Cell number	[-]
$k^{(2)}$	Constant parameter	[-]
$k^{(4)}$	Constant parameter	[-]
M	Mass of the gas part in the sphere	[kg]
M_L	Mass of the liquid part	[-]
$\dot{m}$	Mass flow	[kg/s]
N	Number of nodes	[-]
p, p_p	Pressure, upstream of the damper	[N/m^2]
Q	Volumetric flow	[m^3/s]
Q_P	Volumetric flow at node p	[m^3/s]
Q_0	Initial volumetric flow	[m^3/s]
R	Pressure loss	[N/m^2]
R	Radius	[m]
S	Cell face	[m^2]
S_i	Source terms	[-]

Sign	Sign function	[-]
t, Δt	Time, and time step	[s]
T	Total time for closing the valve	[s]
U	Conservative variable vector	[-]
$\widehat{U}$	Fourier transform of U	[-]
V_0, V	The sphere volume, and the gas volume	[m^3]
α	Runge-Kutta integration coefficient	[-]
ε^2, ε^4	Coefficient of the dissipative terms	[-]
μ	Mass flow coefficient	[-]
γ	Specific heat ratio	[-]
ρ_L	Density of the liquid part	[kg/m^3]
ρ_g	Density of the gas in the damper	[kg/m^3]

Acknowledgments

This research has been supported by the Research Department of Sharif University of Technology.

References

Bellos, C. V., Soulis, J. V. and **Sakkas, J. G.** 1991. Computation of Two-Dimensional Dam-Break Induced Flow. *Advances in Water Resour,* Vol. 14(1), pp. 31-41.

Fennema, R. J. and **Chaudhry, M. H.** 1987. Simulation of One Dimensional Dam-Break Flows. *J. Hydr. Res.,* Vol. 25(1), pp. 25-51.

Garcia-Navaho, P. and **Kahawita, R. A.** 1986. Numerical Solution of the St. Venant Equation with the MacCormack Finite Difference Scheme. *Int. J. Numer. Methods in Fluid,* Vol (6), pp. 507-527.

Garcia-Navaho, P., Priestley, A. and **Alcrudo, F.** 1994. An Implicit Method for Water Flow Modeling in Channel and Pipe. *J. Hydr. Res.,* Vol. 32(5), pp. 721-742.

Jameson, A., Schmidt, W. and **Turkel, E.** 1981. Numerical Solution of Euler Equation by Finite Volume Methods Using Rung-Kutta Time Stepping Schemes. *AIAA 14th Fluid and Plasma Dynamic Conference,* Palo Alto.

Katopodes, N. D. and **Sterlkoff, R.** 1978. Computing Two-Dimensional Dam-Break Flood Waves. *J. Hydr. Div.* ASCE, Vol. 104(9), pp. 1269-1288.

Lai, C. 1979. Comprehensive Method of Characteristics Models for Flow Simulation. *J. Hydr. Engrg,* ASCE, Vol. 114(9), pp. 1074-1097.

Leaf, G. K. and **Chawla, T. C.** 1979. Numerical Methods for Hydraulic Transients. *Numerical Heat Transfer,* Vol. 2, pp. 1-34.

Mingham, C. G. and **Causon, D. M.** 1998. High-Resolution Finite Volume Method for Shallow Water Flow. *J. Hydr. Engrg.* ASCE, Vol. 124(6), pp. 605-614.

Nujic, M. 1995. Efficient Implementation of Non-Oscillatory Schemes for Computation of Free-Surface Flows. *J. Hydr. Res.,* Delft, the Netherlands, Vol. 33(1), pp. 101-111.

Rao, V. S. and **Latha, G.** 1992. A Slope Modification Method for Shallow Water Equation. *Int. J. Methods in Fluid,* Vol. 14, pp. 189-196.

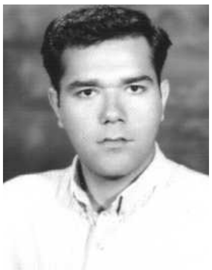
Roe, P.L. 1981. Characteristics-Based Schemes for Euler Equations. *Anal.Rev. Fluid Mech.,* pp. 337-365.

Savic, L. J. and **Holly, F. M., Jr.** 1993. Dam-Break Flood Waves Computed by Modified Godunov Method. *J. Hdry. Res.,* Delft, the Netherlands, Vol. 31(2), pp. 187-204.

Streeter, V. L. and **Wylie, E. B.** 1981. *Fluid Mechanics.* McGraw-Hill Book Co.

Zhao, D. H., Shen, H. W., Tabios, G. Q., Lai, J. S. and **Tan, W. Y.** 1994. A Finite Volume Two-Dimensional Unsteady Flow Modle for River Basins. *J. Hydr.Engrg.* ASCE, Vol. 120(7), pp. 863-883.

Karim Mazaheri
BS, MSc, MA, MS, PhD, was graduated from University of Michigan in 1992. Then he joined to the Sharif University of Technology in Tehran, where still he continues his career. Currently he is an associate professor of aerospace engineering. His main interests include computational fluid dynamics, gas and aerodynamics, combustion engineering, and space systems engineering.



Reza Hatamabadi Farahani
BS, MSc, is a graduate student in the Sharif University of Technology. His main research activity is in the field of computational fluid dynamics, and his speciality is in analysis of hydraulic systems.