



encit 2020



18th Brazilian Congress of Thermal Sciences and Engineering
November 16-20, 2020 (Online)

ENC-2020-0723

COMPUTATIONAL SIMULATION OF TRANSIENT FLOWS WITH CAVITATION

Elias do Carmo Dias

Felipe Sant'Anna Nunes

Mechanical Engineering Department, Federal University of Rio de Janeiro-UFRJ/COPPE, Rio de Janeiro, RJ, Brazil
elias.cd@mecanica.coppe.ufrj.br, fsantannan@gmail.com

Helcio Rangel Barreto Orlande

Mechanical Engineering Department, Federal University of Rio de Janeiro-UFRJ/COPPE, Rio de Janeiro, RJ, Brazil
helcio@mecanica.coppe.ufrj.br

Abstract. *In a transient flow, caused by water hammer, vaporous cavitation can occur when the pressure drops to the vapor pressure of the water. The cavitation process is dangerous and can damage pipelines; therefore, studying and understanding this phenomenon may give us important insights regarding its occurrence and behavior. Throughout the last decades, many studies have been developed in this area due its economic importance. In this work, it is performed the numerical simulation of a one-dimensional transient flow in a straight pipeline generated by a sudden valve closure located at the end of the pipeline. The goal is to predict the pressure peaks at the valve section. For the simulation, a model based on the conservation of mass, conservation of momentum and a transport equation is built. The dependent variables to be evaluated are density, velocity and vapor mass fraction. To compute the pressure, a constitutive relationship is needed. It is assumed that the bubbles are uniformly distributed along the pipeline and the mixture is homogeneous; therefore, liquid and gas have the same velocity. Two distinct numerical methods are used to solve the mentioned problem: the FORCE scheme and the MacCormack method. The results obtained are compared with numerical and experimental results available in literature. The results showed that the MacCormack method produces oscillations, which are expected in a second order, dispersive method. These oscillations are not seen in the FORCE scheme. Nonetheless, it was shown that both methods are accurate in predicting the pressure peaks at the valve section.*

Keywords: *cavitation, water hammer, MacCormack, FORCE scheme*

1. INTRODUCTION

Sudden changes to fluid flow may result in a pressure surge that travel at the speed of sound. This phenomenon is called water hammer and it may be caused by pump starting, pump stopping, pipe rupture, valve opening, valve closing, etc. (Jensen et al., 2018). During the water hammer, when the pressure drops to the dissolved gas saturation pressure or the vapor pressure, the flow regime changes from single-phase to two-phase through the nucleation of bubbles. These phenomena are referred to as cavitation and can be divided in two cases; one is the gaseous cavitation and the other vapor cavitation. The gaseous cavitation occurs when the pressure is below dissolved gas saturation pressure but above the vapor pressure leading to a gas release. The vapor cavitation, known as column separation, occurs when the pressure drops to the vapor pressure of the liquid (Bergant et al., 2006).

The collapse of the vapor bubbles leads to pressure surges that may be higher than water hammer pressure. The cavitation can damage pipelines; therefore, understanding this phenomenon is a big concern of engineering. When cavitation occurs, a degradation is observed in the performance of flow devices, which manifests in different ways such a flow rate fluctuation, load asymmetry, vibrations and noise (Sumam et al., 2009).

In this work, it is performed the numerical simulation of a one-dimensional transient flow in a straight pipeline generated by a sudden valve closure located at the end of the pipeline. The sudden valve closure leads to a distributed vapor cavitation. Two numerical methods are used: The FORCE scheme and the MacCormack method. The results obtained are compared with results available in literature.

2. MATHEMATICAL MODEL

The mathematical model is based on the work of Sumam et al. (2009). This is a one-dimensional model based on the conservation of mass, conservation of momentum and a transport equation. The dependent variables to be evaluated are density, velocity and vapor mass fraction. To compute the pressure, a constitutive relationship is needed.

It is assumed that the bubbles are uniformly distributed along the pipeline, the bubbles are very small in size and are spherical in shape, the pressure is the same across the bubble, vapor bubbles follows a polytropic compression law and that the mixture is homogeneous, i.e., liquid and gas have the same velocity (Sumam et al. 2009).

For a homogeneous mixture, the on dimensional equations of conservation of mass and conservation of momentum can be written respectively by (Wylie and Streeter, 1978):

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} = 0 \quad (1)$$

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(p + \rho u |u|)}{\partial x} + \rho g \sin \theta + \frac{f \rho u |u|}{2D} = 0 \quad (2)$$

in which ρ , p and u are the density, absolute pressure and velocity of the mixture, respectively; D is the internal diameter of the pipe; x is the pipe distance in the direction of the flow; t is the time and f is the friction factor.

In order to describe the cavitation process, the transport equation is used, and it can be written in the following form:

$$\frac{\partial(\rho f_v)}{\partial t} + \frac{\partial(\rho u f_v)}{\partial x} = R_e - R_c \quad (3)$$

in which R_e is the vapor generation and R_c is the vapor collapse.

The equations for R_e and R_c are respectively given by (Singhal et al., 2002):

$$R_e = C_e \frac{\sqrt{k}}{\sigma} \rho_l \rho_v \left[\frac{2}{3} \frac{p_v - p}{\rho_l} \right]^{1/2} (1 - f_v) \quad (4)$$

$$R_c = C_c \frac{\sqrt{k}}{\sigma} \rho_l \rho_l \left[\frac{2}{3} \frac{p - p_v}{\rho_l} \right]^{1/2} f_v \quad (5)$$

in which k is the turbulent kinetic energy given by $\sqrt{k} = 0.1(u)$, ρ_l is the liquid density, ρ_v is the vapor density, σ is the surface tension and p_v is the vapor pressure. C_e and C_c are empirical constants and their value used in this work 0.02 and 1, respectively.

Through the relationship for the liquid density given by Hadj-Taieb et al. (2000) and the relationship for the vapor density given by Coutier-Delgosha et al. (2005) the constitutive equation is written in the following form:

$$\rho = \rho_{i_0} e^{\frac{(p-p_0)}{K_l}} (1 - \alpha_v) + \alpha_v \frac{p}{RT} \quad (7)$$

in which p_0 is the initial pressure, K_l the bulk modulus, R is the gas constant, T is the temperature and α_v is the volumetric fraction of vapor.

The volumetric fraction of vapor varies in each point of the mesh due to the pressure changes along the pipe. This variation can be computed writing (Suman et al. 2009)

$$\alpha_i^{k+1} = \left(\frac{p_i^k}{p_i^{k+1}} \right)^{1/n} (\alpha_i^k) \quad (8)$$

with the Eq. (8), Eq. (7) can be rewritten as:

$$f(p_i^{k+1}) = \alpha_{v_i}^k \left(\frac{p_i^k}{p_i^{k+1}} \right)^{\frac{1}{n}} \frac{p_i^{k+1}}{RT} + \left(1 - \alpha_{v_i}^k \left(\frac{p_i^k}{p_i^{k+1}} \right)^{\frac{1}{n}} \right) \rho_{to} e^{\frac{(p_i^{k+1} - p_o)}{K_i}} - \rho_i^{k+1} = 0 \quad (9)$$

As Eq. (9) is non-linear and differentiable in p_i^{k+1} , the pressure at the next time step can be found using the Newton-Raphson method

$$p_i^{q+1} = p_i^q - \frac{f(p_i^q)}{f'(p_i^q)} \quad (10)$$

where q is the iteration number, $f(p_i^q)$ is the function value after the iteration number q and $f'(p_i^q)$ is the value of the derivative of $f(p_i^q)$ in the given iteration.

Equations. (1), (2) and (3) can be written in the matrix form as:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial x} = \mathbf{S}(\mathbf{U}) \quad (11)$$

where \mathbf{U} is the state vector, \mathbf{F} is the flux vector and \mathbf{S} the source term vector, given respectively by:

$$\mathbf{U} = \begin{pmatrix} \rho \\ \rho u \\ \rho f_v \end{pmatrix} \quad (12)$$

$$\mathbf{F}(\mathbf{U}) = \begin{pmatrix} \rho u \\ p + \rho u |u| \\ \rho u f_v \end{pmatrix} \quad (13)$$

$$\mathbf{S}(\mathbf{U}) = \begin{pmatrix} 0 \\ -\frac{f \rho |u| u}{2D} - \rho g \sin \theta \\ R_e - R_c \end{pmatrix} \quad (14)$$

3. PHYSICAL MODEL

The physical problem consists of a straight horizontal pipe of length L and diameter d . Initially, water (liquid) flows in steady state into the pipeline with a given volumetric flow. Since the upstream pressure is known, it is possible to calculate the initial pressure along the pipe considering the pressure losses due to friction with the walls as (Szydowski, 2002):

$$p = p_{abs} - f \frac{\Delta L}{D} \rho \frac{u_o^2}{2} \quad (15)$$

where p_{abs} is the upstream absolute pressure, u_o is the flow initial velocity, ΔL is the distance between the entrance pipe section and an specific position in the pipe and f is the friction factor.

The transient is generated by means of a linear valve closure located at the downstream, that is, at the end of the pipe. During this hydraulic transient, if the pressure drops to the water vapor pressure, vapor bubbles will occur, characterizing the cavitation process. With the start and end of the valve closure times, it is possible to calculate the decrease in the flow velocity over time as (Szydowski, 2002):

$$u = u_o - u_o \frac{t - tc_i}{tc_f - tc_i} \quad (16)$$

where tc_i and tc_f are the initial and final times of the valve closure, respectively.

At the upstream, the pressure is kept constant and equal to the initial pressure during the simulation. At the downstream, the velocity is calculated with Eq. (1), Eq. (2) and Eq. (3) while the valve is open; the velocity is zero when the valve is closed; and is calculated with Eq. (16) during the valve closure.

An illustration of the physical problem can be seen in Fig. 1.

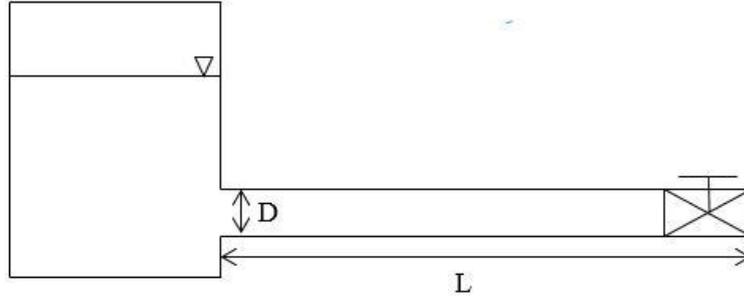


Figure 1. Physical problem illustration

4. SOLUTION METHODS

4.1 Force Scheme

The Force Scheme is a numerical method based on finite volumes, with monotone and first-order scheme. In order to calculate the variables in the state vector it is used the equation given by:

$$\mathbf{U}_i^{k+1} = \mathbf{U}_i^k + \frac{\Delta t}{\Delta x} (\mathbf{F}_{i-1/2}^k - \mathbf{F}_{i+1/2}^k) \quad (17)$$

where Δx is the mesh size, Δt is the time step and $\mathbf{F}_{i+1/2}^n$ is the flux vector between two volumes.

To solve the explicit scheme presented in Eq. (17), the flux at the volume boundaries must be calculated. This is done using the FORCE flux, which is an arithmetic mean of the Lax-Friedrichs flux and Richtmyer flux (Toro, 2009 and Zeidan, 2016). Therefore, one can write:

$$\mathbf{F}_{i+1/2}^{FORCE} = \frac{1}{2} (\mathbf{F}_{i+1/2}^{LF} + \mathbf{F}_{i+1/2}^{RI}) \quad (18)$$

where $\mathbf{F}_{i+1/2}^{FORCE}$ is the FORCE flux, $\mathbf{F}_{i+1/2}^{LF}$ is the Lax-Friedrichs flux and $\mathbf{F}_{i+1/2}^{RI}$ is the Richtmyer flux.

The Lax-Friedrichs flux and the Richtmyer flux are given, respectively, by:

$$\mathbf{F}_{i+1/2}^{LF} = \frac{1}{2} [\mathbf{F}(\mathbf{U}_i^k) + \mathbf{F}(\mathbf{U}_{i+1}^k)] + \frac{1}{2} \frac{\Delta x}{\Delta t} [\mathbf{U}_i^k - \mathbf{U}_{i+1}^k] \quad (19)$$

$$\mathbf{F}_{i+1/2}^{RI} = \mathbf{F}\left(\frac{1}{2}(\mathbf{U}_i^k + \mathbf{U}_{i+1}^k)\right) + \frac{1}{2} \frac{\Delta t}{\Delta x} [\mathbf{F}(\mathbf{U}_i^k) - \mathbf{F}(\mathbf{U}_{i+1}^k)] \quad (20)$$

The time step was calculated using the criteria proposed in Toro (2009)

$$\Delta t = C_{cfl} \frac{\Delta x}{S_{\max}^k} \quad (21)$$

where C_{cfl} is Courant-Friedrichs-Lewy coefficient and S_{\max}^k is the fastest wave speed along the grid at the instant time k .

4.2 MacCormack Method

The numerical method of MacCormack (1969) was used for the solution of the cavitation problem given by Eq. (1), Eq. (2) and Eq. (3). The method consists of approximating the vectors in Eq. (11) by means of finite difference method in its explicit form through two steps, prediction and correction. In the prediction step, the spatial derivatives are replaced by forward finite differences and in the correction step, those are replaced by backward finite differences. In this way:

Prediction

$$\mathbf{U}_i^{\overline{k+1}} = \mathbf{U}_i^k - \frac{\Delta t}{\Delta x} (\mathbf{F}_{i+1}^k - \mathbf{F}_i^k) + \Delta t \mathbf{S}_i^k \quad (22)$$

Correction

$$\mathbf{U}_i^{k+1} = \frac{1}{2} \left[\mathbf{U}_i^k + \mathbf{U}_i^{\overline{k+1}} - \frac{\Delta t}{\Delta x} (\mathbf{F}_i^{\overline{k+1}} - \mathbf{F}_{i-1}^{\overline{k+1}}) + \Delta t \mathbf{S}_i^{\overline{k+1}} \right] \quad (23)$$

The prediction step creates approximate values $\mathbf{U}_i^{\overline{k+1}}$ for \mathbf{U}_i^{k+1} in each grid point. With those approximate values, $\mathbf{F}_i^{\overline{k+1}}$ and $\mathbf{S}_i^{\overline{k+1}}$ are calculated. The solution \mathbf{U}_i^{k+1} is then obtained with the correction step.

As this is an explicit method, the maximum time step must satisfy the Courant-Friedrichs-Lewy (CFL) criteria. Therefore, the time step for each iteration can be obtained with Eq. (24). However, for the cavitation phenomenon to be properly portrayed in this case, it is necessary to use time steps smaller than 10^{-6} s.

$$\Delta t^k = \min \left(\frac{\Delta x}{|u_i| + a_i} \right) \quad (24)$$

For the problem to be well posed, the method requires that some variables should be specified at the boundaries and extrapolated to fictitious nodes. Denoting x as the specified variable and subscript denoting the grid point, it can be written that $x_0 = x_1$ and $x_L = x_{L-1}$. At the pipe inlet, pressure and velocity are always specified. At the pipe outlet, while the valve is open, the pressure is specified and after the valve closure, the velocity is specified.

For being a second order method, it presents oscillations near regions of discontinuity and introduces dispersion in the numerical solution. To reduce this effect, it is common to introduce an artificial viscosity term. This term is large in regions where the second derivative is large (regions of discontinuity) and small where the solution is smooth. The artificial viscosity model used in this work as proposed by MacCormack and Baldwin (1975) and is known to be efficient in reducing oscillations without causing much dissipation in the numerical solution. The artificial viscosity term is introduced into the flux \mathbf{F} and is given by:

$$\mathbf{M} = -\varepsilon \Delta x^3 \frac{|u| + a}{4p} \left| \frac{\partial^2 p}{\partial x^2} \right| \frac{\partial \mathbf{U}}{\partial x} \quad (25)$$

The pressure derivative is replaced by centered finite differences, while the \mathbf{U} derivative is replaced by finite differences in the opposite direction of flux \mathbf{F} . The artificial viscosity term in the prediction and correction steps becomes, respectively:

$$\mathbf{M}_i^k = -\varepsilon (|u| + a)_i^k \frac{|p_{i+1}^k - 2p_i^k + p_{i-1}^k|}{p_{i+1}^k + 2p_i^k + p_{i-1}^k} (\mathbf{U}_i^k - \mathbf{U}_{i-1}^k) \quad (26)$$

$$\mathbf{M}_i^{\overline{k+1}} = -\varepsilon (|u| + a)_i^{\overline{k+1}} \frac{|p_{i+1}^{\overline{k+1}} - 2p_i^{\overline{k+1}} + p_{i-1}^{\overline{k+1}}|}{p_{i+1}^{\overline{k+1}} + 2p_i^{\overline{k+1}} + p_{i-1}^{\overline{k+1}}} (\mathbf{U}_{i+1}^{\overline{k+1}} - \mathbf{U}_i^{\overline{k+1}}) \quad (27)$$

Adding the viscosity terms given by Eq. (27) and Eq. (26) in Eq. (22) and Eq. (23), prediction and correction steps becomes, respectively:

$$\mathbf{U}_i^{\overline{k+1}} = \mathbf{U}_i^k - \frac{\Delta t}{\Delta x} (\mathbf{F}_{i+1}^k + \mathbf{M}_{i+1}^k - \mathbf{F}_i^k - \mathbf{M}_i^k) + \Delta t \mathbf{S}_i^k \quad (28)$$

$$\mathbf{U}_i^{k+1} = \frac{1}{2} \left[\mathbf{U}_i^k + \mathbf{U}_i^{\overline{k+1}} - \frac{\Delta t}{\Delta x} (\mathbf{F}_i^{\overline{k+1}} + \mathbf{M}_i^{\overline{k+1}} - \mathbf{F}_{i-1}^{\overline{k+1}} - \mathbf{M}_{i-1}^{\overline{k+1}}) + \Delta t \mathbf{S}_i^{\overline{k+1}} \right] \quad (29)$$

5. RESULTS

The test cases presented was taken from Suman et al. (2009). In this section, the results obtained with the FORCE scheme and with the MacCormack method for the pressure at the valve section will be compared. The summary of the information regarding the problem can be seen in Tab. 1. The information regarding the methods used can be seen in Tab. 2. Two distinct cases were studied. The difference between them is in the value of the initial volumetric flow rate, which can be seen in Tab. 3 for each case.

Table 1. Information regarding the proposed problem

Upstream absolute pressure	p_{abs}	267,000	(Pa)
Pipe length	L	32.5	(m)
Pipe diameter	D	0.053	(m)
Pipe wall thickness	e	0.0037	(m)
Pipe Young's modulus	E	200×10^9	(Pa)
Friction factor	f	0.0302	(-)
Temperature	T	22	(C)
Polytropic coefficient	n	1.25	(-)
Initial valve closure time	tc_i	0.0	(s)
Final valve closure time	tc_f	0.02	(s)

Table 2. Information regarding the used methods

	Number of grid points	Δx (m)
MacCormack	66	0.5
FORCE	2560	0.013

Table 3. Initial velocity and volumetric flow rate in each test case

		Case 1	Case 2	
Initial velocity	u_o	0.55	0.327	(m s ⁻¹)
Initial volumetric flow rate	Q_o	0.000562	0.00072	(m ³ s ⁻¹)

The results obtained for the pressure at the valve section for case 1 and case 2, shown in Tab. 3, can be seen in Fig. 2 and Fig. 3, respectively.

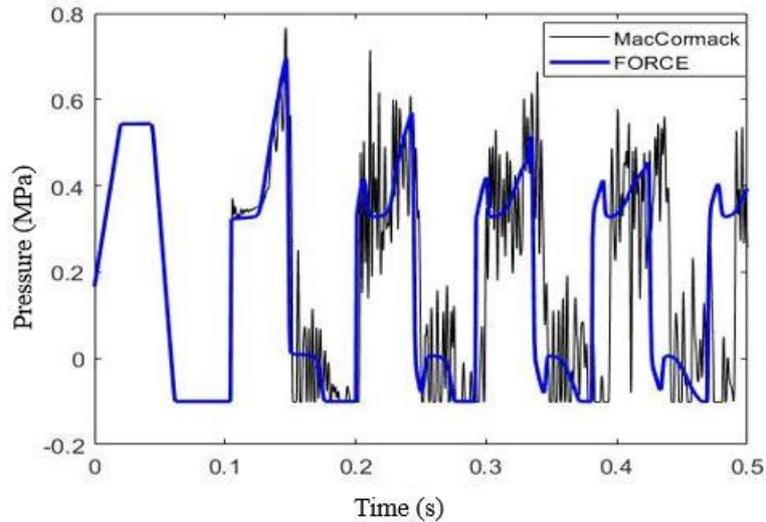


Figure 2. Pressure at the valve section obtained with the MacCormarck method and with the FORCE scheme for case 1

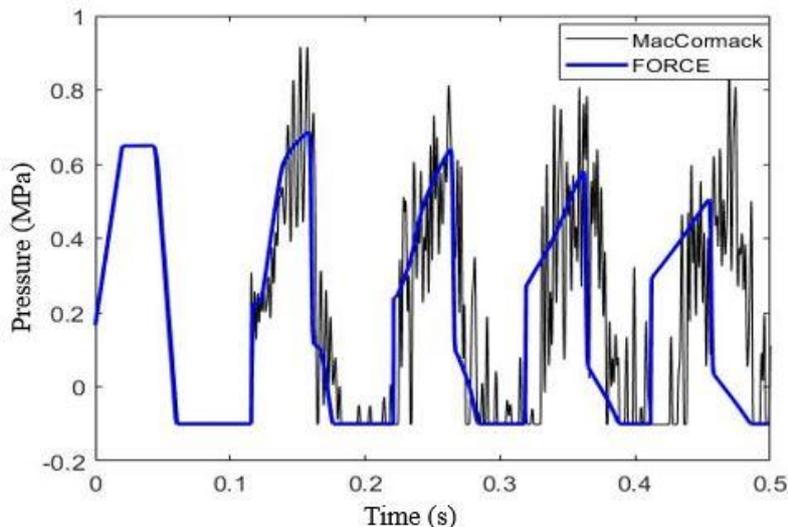


Figure 3. Pressure at the valve section obtained with the MacCormarck method and with the FORCE scheme for case 2

Initially, the MacCormack method was applied without using the artificial viscosity term. In this case, there is a large dispersion of the solution in the vicinity of the discontinuities. Such effect is already expected in second-order methods. The use of the artificial viscosity tends to introduce dissipation in regions of discontinuity; however, its value must be adjusted so that these regions are not degenerated. On the other hand, the FORCE scheme shows better stability near the regions of discontinuity when compared with the MacCormack method, producing a better representation of the water hammer.

The calculation of the time step for the MacCormack method is based on the Courant-Friedrichs-Lewy (CFL) stability criteria. As the wave speed drops dramatically, the time step changes from 10^{-4} to 10^{-2} s. However, the vapor collapse can occur in much smaller time steps and not captured. For this reason, the time step used in these regions was reduced and kept in the order of 10^{-7} s. For the FORCE scheme, the time step is calculated with Eq. (21), varying with de mixture celerity and with the cell length. For a mesh with 2560 volumes, the time step was found to be on the order of 10^{-6} s.

In the results presented in Fig. 2 and Fig. 3 is noted that the pressure in the valve section reaches the first peak value due to the water hammer phenomenon. Subsequently, the pressure is reduced to a value below the atmospheric pressure, close to the vapor pressure region. This pressure value is maintained, generating vapor, until the next high pressure cycle occurs. As the pressure increases in the next cycle, the generated vapor collapses, which can result in pressure spikes greater than the spike resulting from water hammer.

In the peaks following the first pressure peak, oscillations are noted in its ascending and descending portions, which is not commonly seen in pressure waves resulting from the water hammer. This is the first indicator of the collapse of vapor. Another indicator is seen in the calculated wave speed. Its variation is going from the order of 10^3 to 10^1 m^{-1} in a

short period of time, indicating a great variation in the water compressibility module, which can only occur if there is generation and collapse of vapor.

In Suman et al. (2009), the MacCormack method was also used to generate the solutions for the pressure in the valve section. Therefore, the presence of noise in the solution is noted, similar to the behavior presented in the solution obtained through the MacCormack method implemented in this work.

6. CONCLUSION

In this work, a one-dimensional model based on the continuity equation, momentum equation and a transport equation were built to simulate fluid transients generated by means of a linear valve closure, located at the downstream of the system.

Two numerical methods were implemented in order to evaluate the pressure peaks and the cavitation occurrence over time in the valve section due to the water hammer phenomenon. Those were: the MacCormack method and the FORCE scheme.

The results showed that the solution obtained with the MacCormack method produced oscillations near the discontinuities due to the dispersive effect produced by this second order method. On the other hand, the FORCE scheme produced stable solutions near the discontinuities. Overall, from the comparison between the results obtained in this work and the results presented in Suman et al. (2009), it can be concluded that the model and both methods implemented in this work represent properly the water hammer phenomenon with cavitation occurrence, as well as the maximum pressure value for this phenomenon.

7. REFERENCES

- Bergant, A., Simpson, A.R. and Tijsseling, A.S., 2006. "Water Hammer with Column Separation: A Historical Review". *Journal of Fluids and Structures*, Vol. 22, No. 2, pp. 135-171.
- Coutier-Delgosha, O., Fortes-Patella, R., Reboud, J.L., Hakimi, N. and Hirsch, C., 2005. "Stability of Preconditioned Navier Stokes Equations Associated with a Cavitation Model". *Computers and Fluids*, Vol. 34, pp. 319-349.
- Hadj-Taieb, E. and Lili, T., 2000. "Validation of Hyperbolic Model for Water Hammer in Deformable Pipes". *Journal of Fluids Engineering*, Vol. 122, No. 1, pp. 57-64.
- Jensen, R.K., Larsen, J.K., Lassen, K.L., Mando M. and Andreasen A., 2018. "Implementation and Validation of a Free Open Source 1D Water Hammer Code". *Journal of Fluids*, pp 3-64.
- MacCormack, R.W., 1969. "The Effect of Viscosity in Hypervelocity Impact Cratering". *AIAA Paper* 69-357.
- MacCormack, R.W., Baldwin, B., 1975. "A Numerical Method for Solving the Navier-Stokes Equations with Application to Shock Boundary Layer Interactions". *AIAA Paper* 75-1.
- Sinhal, A.K., Athavale, M.M., Li, H. and Jiang, Y., 2002. "Mathematical Basis and Validation of Full Cavitation Model". *Journal of Fluids Engineering*, Vol. 124, pp. 617-624.
- Sumam, K.S., Thampi, S.G. and Sajikumar, N., 2009. "An Alternate Approach for Modelling of Transient Vaporous Cavitation". *Journal for Numerical Methods in Fluids*, Vol. 63, pp. 564-583.
- Szydlowski, M., 2002. "Finite Volume Method for Water Hammer Simulation". In: *International Scientific and Technical Conference on Technology, Automation and Control of Wastewater and Drinking Water Systems*. TiASWiK'02, 159-165.
- Toro, E.F., 2009. "*Riemann Solvers and Numerical Methods for Fluid Dynamics*". Springer-Verlag, Berlin, Heidelberg.
- Wylie, E. B and Streeter, V.L., 1978, "*Fluid Transients*", McGraw-Hill, New York.
- Zeidan, D., 2016, "Assessment of Mixture Two Phase Flow Equations for Volcanic Flow Using Gudunov Type Methods". *Applied Mathematics and Computation*, Vol. 272, pp. 707-719.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.