Comparing Pure CFD and 1-D Solvers for the Classic Water Hammer Models of a Pipe-Reservoir System

Sharon Mandair¹, Bryan Karney¹, Robert Magnan², Jean-François Morissette²

¹ University of Toronto, 35 St George St, Toronto, ON, Canada, M5S 1A4
² Institut de recherche d’Hydro-Québec, 1800, boul. Lionel-Boulet, Varennes, QC, Canada, J3X1S1

¹sharon.mandair@mail.utoronto.ca

ABSTRACT
Various system representations of transient events are possible, each with respective strengths and weaknesses. This paper compares results from a water hammer experiment (courtesy of Bergant et al.) to a set of simulations using a series of numerical models. The experimental rig includes two tanks connected by a long pipe under laminar flow. The water hammer event is caused by the rapid closure of a terminal valve. The transient flow is computed by means of two solvers. The first one is a basic one-dimensional (1D) Method of Characteristics (MOC) solver including unsteady friction terms based on instantaneous acceleration. The second is a 3D commercial CFD solver of the pipe. The 3D pipe model captures part of the damping caused by unsteady wall friction, but can artificially capture the pressure signal’s phase shift. The challenges involved in setting up the CFD models are discussed in detail. The longer-term goal of this work is to attempt more complex 1D-3D couplings, but there are notable challenges even in simple cases.

Keywords: water hammer, Method of characteristics, CFD

1 Introduction

With the wide range of transient flow models on offer, the challenge for the user is to decide what level of complexity is needed and what cost is worth paying to capture this complexity. Among the simple 1D models, the Method of Characteristics (MOC) is the most common. This interpretation of pressure wave propagation is well suited for pipelines and networks. The MOC equations numerically solve the continuity and momentum equations along characteristic lines, described by the wave speed. Using a simple set of equations (provided in detail in Section 2), it can be extended to include additional features, such as cavitation and unsteady friction [1]. More detailed and computationally expensive models include 2D and 3D CFD models, based on the Navier Stokes equations. Water hammer can be simulated with the addition of a compressibility model.

As more sophisticated 3D models become widespread, it is practical to couple them with simple 1D models. In fact, 1D-3D coupling is the long-term goal for the authors, with the aim to evaluate several coupling configurations applied to a 1D-3D model of Bergant’s experimental rig. Of the numerous coupling approaching in the literature, the authors are interesting in non-iterative methods such as the straight and overlapped schemes presented in [2] and [3]. The straight coupling interface is simple to implement but risks introducing spurious transients because it misaligns the time-steps of the two models. To address this issues, Zhang and Cheng suggest overlapping the two models; depending on the time step, this may require a great deal more resources from the CFD side. Applying these approaches to a simple pipe line will allow the authors to evaluate each interfaces before introducing a more complex CFD model.
The present work is in preparation for the coupling evaluation since it is useful to first understand how each of these models handle the same case. In this article, the authors compare different model representations of a water hammer event in a small experimental set up. The authors discuss how the models affect both damping and wave speed, and anticipate potential challenges in coupling MOC and CFD models.

1.1 Experiment description

Bergant's 1990 experimental rig is a 22-mm pipe, 37 m in length, book-ended by two reservoirs. Bergant et al. recorded the pressure at five points along the pipe after a sudden downstream valve closure. A schematic of the rig is shown in Figure 1. The purpose of the original work was to evaluate the Zielke and Brunone unsteady friction models, which are built into an MOC model. The experiment was performed with a variety of configurations and initial flow conditions, but the present work focuses on a downstream valve closure with an initial flow velocity of 0.1 m/s. This is a laminar flow case, where the Reynolds number is 1870 [4].

![Figure 1: Bergant et al.'s experimental rig](image)

The reservoirs on either end were roughly 0.5 m in diameter, and 2 m tall but maintained a hydraulic head around 32 m with the help of compressed air. The air pressure was governor controlled, reading and responding to the head measured at the bottom of the tank.

Given the minuscule flow rate ($3.8 \times 10^{-5} \text{ m}^3/\text{s}$) and the short experiment time (1.5 s), a feedback line was unnecessary. To initialize the experiment, the valve was opened and the water level in the tank allowed to drop until the flow velocity reached the desired magnitude, at which point the valve was closed rapidly ($t_c = 9 \text{ ms}$) to produce the water hammer effect.

For the present work, this version of Bergant et al.'s experiment is recreated with a variety of models, and the results will be compared to their experimental results. The authors of the experiment concluded in later work that though the Zielke model is costlier than the Brunone model, it is advantageous in certain pipeline configurations. For the configuration of interest in this research, however, the Brunone method performs equally [4].
2 Model descriptions

The Bergant experiment is simulated with the following four models.

1. 1D Method of Characteristics (MOC)
2. 1D MOC with local loses
3. 1D MOC with Brunone friction model
4. 3D CFD of the pipe

2.1 Method of Characteristics

The Method of Characteristics (MOC) is a numerical approach to solving partial differential equations (PDE). In the case of transient flow, the PDE's for conservation of mass and momentum are resolved along characteristic lines, described by the wave speed. The fundamental equations are shown below, and they are solved using the scheme presented by Karney and McInnis [5].

\[
\frac{\delta H}{\delta t} + a^2 \frac{\delta V}{g \delta x} = 0 \tag{1}
\]

\[
\frac{\delta H}{\delta x} + \frac{1}{g} \frac{\delta V}{\delta t} + f\frac{|V|}{2D} = 0 \tag{2}
\]

\[
\pm a = \frac{dx}{dt} \tag{3}
\]

where \( H \) is head, \( V \) is the axial flow velocity, \( a \) is wave speed, and \( g \) is gravitational acceleration, \( f \) is the friction factor, and \( D \) is the pipe diameter. Effectively, the flow conditions at some point, \( P \) at time \( t \) and position \( x \), can be resolved using flow and head data from two points \( A \) and \( B \), at \((x-1, t-1)\) and \((x+1, t-1)\), respectively.

The valve in the MOC model was treated as an external energy dissipator (EED) as presented by Karney and McInnis [5]:

\[
Q_{ext} = s \tau E_s \sqrt{s \Delta H_P} \tag{4}
\]

where \( Q_{ext} \) is the flow exiting the node, \( s \) is the sign of the flow term, \( \tau \) is the valve closure term, \( E_s \) is the orifice parameter, and \( \Delta H_P \) is the head loss across the EED. For the second model where the pressure wave is attenuated through local loses, an EED is placed at the upstream node, the characteristics for which are chosen to match the experimental results.

2.1.1 Brunone friction model

As presented above, the MOC uses the steady-state friction, typically seen in the Darcy Weisbach equation, but this is inadequate for small diameter pipes, where shear stresses are significant. To include transient friction, \( f \) can be redefined as the sum of steady and unsteady components, as in Equation 5. During a transient event, velocity profiles change with the passage of the wave front, resulting in high velocity gradients near the wall, with respect to both time and position [6].

Brunone's model for unsteady friction (Equation 6) aims to correct for this. It is based on instantaneous acceleration and is simple to implement since the unsteady component \( (f_u) \) is defined using the same local acceleration and convection terms as in the MOC model. This model introduces one new term, \( k \), the Brunone friction coefficient. Vardy and Brown relate this term to the Reynolds number as shown in Equations 7-9, where \( Re \) is Reynolds number [4].

\[
f = f_s + f_u \tag{5}
\]

\[
f_u = \left( \frac{kD}{\sqrt{\left| V \right|}} \right) \frac{\delta V}{\delta t} - a \text{ sign}(V) \left( \frac{\delta V}{\delta x} \right) \tag{6}
\]

\[
k = \sqrt{C^*} \tag{7}
\]

Laminar flow: \( C^* = 0.00476 \)
Turbulent flow: $C^* = \frac{7.41}{R_{e^{\log(14.3/Re^{0.05})}}}$  

(9)

As explained by Vitkovsky et al., this formulation can be understood in terms of its impact on the damping and frequency of the pressure wave. The former is only affected by the convection term $(\delta V/\delta x)$, and the latter by the acceleration term $(\delta V/\delta t)$. Effectively, the addition of the $k$ term reduces the wave speed by a factor of $1+k$, and reduces the pressure head by a factor of $(1+k)^2$ each cycle [7], [6]. Given the independence of the two terms, they can each be given a unique $k$ friction coefficient, as developed by Abreu and Aleida, though their formulation requires a 2D model [8] [9]. In this work, however, a single $k$ value is used.

2.2 CFD Model

The CFD model uses the Navier-Stokes equations to compute the flow in a 3D mesh of the whole pipe. Equations 10 and 11 represent continuity and momentum, respectively).

$$\frac{\delta \rho}{\delta t} + \nabla \cdot \rho \vec{V} = 0$$

(10)

$$\frac{\delta \rho \vec{V}}{\delta t} + \nabla \cdot (\rho \vec{V} \vec{V}) - \nabla \cdot \mu \nabla \vec{V} = -\nabla P$$

(11)

where $\rho$ is density, $\vec{V}$ is velocity, $\mu$ is viscosity, and $P$ is pressure.

The structured, multi-block mesh is shown in Figure 2, with contour lines depicting the steady state velocity profile at the midpoint of the pipe. Axially, it is made up of 250 reaches, with $13.3 \times 10^3$ nodes in each cross section. The total mesh size is $3.316 \times 10^6$ nodes. The mesh element neighbouring the wall has a thickness of 0.2 mm.

The reservoir and the valve are not actually modelled, but simply replaced by appropriate boundary conditions. For the valve, this is an average velocity at the outlet, which is reduced linearly to zero. The upstream reservoir is represented by an opening at constant pressure.

![Figure 2: CFD mesh. Elements are 0.149 m long in the axial direction](image)
2.2.1 Compressibility model

To capture the pressure wave propagation, which is a result of fluid compressibility, the fluid density is modelled as a function of pressure. In the equation below $\rho$ is the compressed density, $\rho_l$ is the incompressible density at atmospheric pressure, $a$ is the same wave speed used in the MOC model, $P$ is static pressure, and $P_{\text{atm}}$ is atmospheric pressure. Here, $a$ is set to 1319 m/s, the theoretical wave speed, adjusted for pipe wall deformation, as presented by Wylie and Streeter [10]. Consequently, this model of wave propagation neglects the fluid-structure interaction, maintaining a constant cross-sectional area.

$$\rho = \rho_l + a^{-2}(P - P_{\text{atm}})$$ (12)

The treatment of density differs between MOC and CFD. Though compressibility is foundational to the MOC model, density is assumed to be constant since the change in density is negligibly small. The change in pressure is the dominant manifestation of compressibility.

2.2.2 Friction model

Using the no slip wall condition, wall shear is calculated using Equation 13 below, where $\tau_w$ is the shear stress, $\mu$ is the dynamic viscosity, $v$ is the velocity, and $n$ is the normal distance from the wall. This is notably different from the Brunone model, since CFD uses the radial velocity gradient, which is not available in a 1D model.

$$\tau_w = \mu \left( \frac{\partial v}{\partial n} \right)$$ (13)

3 Results and discussion

Figure 3 compares the experimental measurements to results of the three MOC representations of friction:

- Steady friction (Darcy-Weisbach, $f = 0.0345$);
- Steady friction with local losses imposed at the upstream reservoir;
- Brunone’s unsteady friction model.

The differences appear minor at first, but become more evident after several cycles. Including local losses does represent the wave attenuation, however the frequency is unaffected. The unsteady friction model affects the shaping, timing and attenuation. The frequency of the wave does reduce, but never to the extent observed in the experimental results. It is always roughly half way between experimental results and the steady friction model. Furthermore, it overestimates attenuation, as is evident after a dozen cycles.

3.1 3D results

The CFD results align more closely to the MOC steady friction cases. As shown in Figure 4, there is some minor attenuation, and no change in the frequency. In CFD, the mechanism for damping is the wall shear, which is calculated radially, as opposed to axially in Brunone’s approach. After 5 cycles, the amplitude decays by roughly 15%, which is the equivalent to a Brunone $k$ of 0.016, less than half the value used in the unsteady MOC model. Regardless, in comparison to the experimental results, this is a reasonable representation; as described earlier the Brunone model overestimates damping.

On the other hand, frequency and wave shape are unaffected by the shear stress in CFD model. The frequency is independently governed by the compressibility model, which artificially conforms to the given wave speed. So, the 3D model would be improved by using the observed wave speed based on the experimental data rather than the theoretical value used here. The chosen wave speed is estimated.
based on pipeline characteristics whereby the true value in an infinite fluid is corrected for the pipe wall deformation and axial stress caused by the pipe anchors [10]. Since the CFD model maintains a fixed volume, these local deformation dynamics of the wave front are not captured.

In recent work by Martins et al., the 1D unsteady friction models are evaluated with CFD [11]. According to the authors, a higher resolution mesh especially near the wall is necessary to capture the shear layer. Martins et al. used a 2D axisymmetric mesh, uniformly dividing the radius into 65 cells, compared to 32 radial cells in the present case. Axially Martins et al. have a mesh 10 times more refined, however their acoustic Courant number was not reported. It is likely the mesh is not refined enough to capture the viscous force near the wall.

Overall, the present CFD model behaves most similarly to an MOC model with local losses. For the purposes of evaluating coupling methods, this pairing would be the most straightforward, eliminating extraneous issues, but foreclosing any meaningful comparison with experimental data. Alternatively, following the example set by Martins et al, the mesh could be further refined especially in the near-wall region. Hopefully, this will improve the CFD performance, making it more compatible with the 1D transient friction models.

When coupling the two models, a possible configuration will employ average velocity over a cross section either at a boundary (for straight coupling) or a short distance away (overlapped). The CFD model appears to be well suited for this. Figure 5 a and b shows velocity profiles at two cross sections: one at the entrance, and the second 2 m downstream, where flow is fully developed. Figure 5 c and d compares average velocity along the pipe axis in both models. Data is shown at several instances as the wave front is reflected at the reservoir during the first cycle. $t_c$ denotes the time at which the wave front arrives at the reservoir ($L/a$), and $t_c$ is the duration of the valve closure (9 ms). The figures show average velocity is equal at the two cross sections location, and so the coupling interface location should have no negative consequences. That said, the type of boundary condition is consequential. For instance, in Ansys CFX, a velocity boundary imposes a constant velocity profile, whereas mass flow boundary imposes a parabolic profile.

Finally, one of the challenges in building the CFD model are the oscillations in the wake of the wave front. In the model presented here, they are minor but still visible in Figure 4. These appear to be a numerical issue stemming from discontinuities in the velocity field. What is more worrisome is they appear to amplify each cycle. The authors found the oscillations to be exacerbated with a small Courant number ($<0.5$), and minimized by smoothing the valve closure function. Of course, the latter option is not useful when it comes to coupling the MOC and CFD models.

![Figure 3: 1D model results](image-url)
4 Conclusions

For the purposes of evaluating coupling methods, a simpler case is likely more expedient. Unsteady shear introduces several complications, like the necessity for a highly refined mesh in CFD. Nonetheless, this investigation has been useful in understanding the wave behaviour in both CFD and MOC models. In CFD, the shear force has no effect on the wave frequency; which must be artificially tuned using the compressibility model. This is a significant consideration for future coupling work. The two models should treat the pressure wave similarly to avoid spurious transients or interference originating from the coupling interface.

After further refining the mesh to better represent the wall shear stress, future work will look at including the upstream reservoir to see its impact on the wave behaviour. Subsequently, the MOC and CFD models will be coupled, in various configuration to evaluate coupling approaches.

![Figure 4: CFD, MOC and experimental results at midpoint](image)

![Figure 5: Velocity profiles during wave reflection at a) the entrance and b) 2 m further downstream; Average velocity vs axial distance c) within 2r of entrance and d) the first 20 m of the pipe. X denoting CFD, line denoting MOC](image)
5 References


