**Dynamic behaviour of a pipe system under unsteady flow and structure vibration**

M Simão¹, J Mora², H M Ramos¹

¹Departamento de Engenharia Civil, Instituto Superior Técnico, Avenida Rovisco Pais 1, 1049-001 Lisboa, Portugal
²Departamento de Ingeniería Hidráulica, Universidad de Guanajuato, Sede Belén: Av. Juárez 77, Zona centro Guanajuato, Gto. C.P. 36000, Guanajuato, México
email: m.c.madeira.simao@ist.utl.pt, jesusmora@ugto.mx, hr@civil.ist.utl.pt

ABSTRACT: In this article the subject fluid structure interaction is approached using a real and a computational model. Different scenarios are tested in a laboratory facility, where analysis of transient phenomena are created. Since it’s a fluid-structure elastic phenomenon, it’s important to study the phenomenon unsteady flow induced that reflects a structural vibration, developed when the steady-state velocity of a fluid is suddenly altered. Thus, the most adequate approach to model water-hammer phenomena is the one based on CFD (Computational Fluid Dynamics), and the FEM (Finite Element Method) to study the structural characteristics involved in the analysis of under pressure hydraulic structures pipe system. Through these models, it’s possible to study the behaviour of the fluid and the structure, as a whole, i.e. enabling the response of the structure change by the flow modification and, consequently, the response of the fluid by structure vibration. In general, simulations based on CFD analyses are used to better understand the dynamic phenomena. As dealing with a joined problem, two domains are created, the fluid-flow and the structure. The analysis are developed in order to show the possible consequences in terms of displacements, when the system is submitted to different loads and type of supports. It is equally emphasized the importance of integrated analysis (fluid-structure interaction) and the purpose of supports in water pipe systems in the infrastructures design.

KEY-WORDS: fluid-structure interaction; FEM; Water hammer; CFD.

1 INTRODUCTION

Water hammer and surge are commonly occurring phenomena in pipe systems, inducing extreme pressure events, leaks ruptures or even dangerous accidents that are rarely well understood due to the numerous variables associated. The flow through a pipe does not seem likely to be able to generate high pressures, or create a risk to joints and mountings, yet the effects of water hammer can be to burst joints and break mountings with relative ease. Water flowing through a length of pipe gains a momentum, induced by the water column inertia, and the rapid closing of a valve requires the dissipation of the excess energy. The energy exerts itself in the form of a momentary increase in pressure above the normal initial pressure. If the valve closes very rapidly the pressure rise can lead to the effect of water hammer, where a wave of flow energy travels within the pipe at the celerity speed, creating potential damage [1].

This research is an ongoing study about hydraulic transients using different numerical methods, comparing them with the experimental tests in a laboratory network. The range of results from the test study of hydraulic transients presented in this research highlights the importance of accurate prediction of the pressure variations in order to ensure that a pipeline’s integrity is not compromised.

2 FUNDAMENTALS

Hydraulic transients in closed pipelines have been a subject of theoretical and practical research. A common and simple example is the knocking sound or hammering noise which is often heard when a valve is rapidly closed. The transient state of the flow from time of closure until a new steady state condition is established, is complex due to pressure surges that propagate away from the valve. By closing the valve rapidly, the valve converts the kinetic energy carried by the fluid particles into strain energy in the pipe walls [2]. This results in a “pulse wave” of abnormal pressure to travel from the disturbance into the pipe system. The hammering sound that is sometimes heard results from the fact that a great portion of the fluid’s kinetic energy is converted into pressure waves, causing noise and vibrations in the pipe. Energy losses due to mainly friction cause the transient pressure waves to decay until a new steady state is established [3].
During transient events, the maximum pressures can cause damage in pipelines, valves, or other equipment. Sometimes, high pressures may not necessarily destroy pipelines or other devices, but can cause cracks in internal linings, damage connections between pipes or cause deformations in equipment. This equipment could be valves, air valves, or even hydraulic transient protection devices. Moreover, high pressures may result in leakages in hydraulic systems even though no visible damage can be noticed.

Strong hydraulic vibrations may damage pipeline, internal lining, or system equipment. Such long-term moderate surges may gradually lead to fatigue failure. Oscillations of water masses through a pipeline may also cause vibrations and suction of air into the pipeline. Therefore, neglecting such influences during the design phase may lead to system damage. Also, interaction between fluid-structure manifests itself as vibration behaviour and in disturbances in the velocity and pressure variations of the fluid. The additional pressure loads are transmitted to the mechanical supports which may damage them inducing dangerous dynamic instabilities [4].

The elasticity of the pipe boundaries and the compressibility of the fluid prevent these sudden changes in pressure from taking place instantaneously throughout the fluid. The associated pressure changes during a transient period are often very large and occur very rapidly (within a few seconds). If the maximum pressures exceed the bar ratings (mechanical strength) of the piping material, different types of failure such as pipe bursts can occur. Similarly, if the minimum pressure drops below the vapour pressure of the fluid, cavitation can occur and can be detrimental to the pipeline system [5].

3 DESCRIPTION OF THE LABORATORY FACILITY

In order to study the effects of the fluid structure interaction (FSI), several tests were carried out in the laboratory of Guanajuato’s University. The following figure shows the installation and all components involved. This system comprises a pump whose characteristic curve was determined by assigning different values of flow and pressure. This pumping system supplies the entire piping network. The distance between the surface of the liquid in the supply tank and the centreline of the pump is negative, so there is a suction lift, and then the water accumulated in the deposit tank is raising up to a maximum head of 30 mwc. The maximum discharge of this installation is about 25 L/s. From upstream to downstream, 4 pressure transducers were installed throughout the facility, 1 relief valve installed immediately downstream of the pump and also 2 flow meters set as Figure 1. The first flow meter (Figure 1) corresponds to an ultrasonic. This needs to be installed about 1/8 of the pipe, so that the reading can be more accurate. However, this type of accessory only works accurately for high flow rates, due to the location of their sensors to the flow volume. Thus two flow meters were used, an ultrasonic and a conventional flow meter, also located in the GI pipe.

Figure 1: Laboratory of Guanajuato’s University (scheme).
The system equipment comprises two types of pipes, galvanized iron (GI) and PVC, with lengths of 30 and 85 m, respectively. Regarding the number and type of fittings, GI pipes accounts: 3 Flow control valves (FCV), 1 check valve (CV) of no-flow control, 1 pump, and a discharge tank. As for PVC pipe, this includes 1 butterfly valve, 1 FCV, and a discharge channel.

The pipes are supported on iron supports approximately 0.5 m height, being attached to them through iron or metal clamps, depending on the type of the piping material (iron - GI, metal – PVC).

Regarding the type of joints, these are elastic, but in case of GI pipe the joints are made through flange couplings, and in case of PVC pipe, by Gibault joints.

3.1 Experimental tests

In the configuration for water hammer tests, the pipes are connected to a constant head of 30 m, varying the discharge. The pump provides the water supply to the constant head inlet. The constant head means that water enters the 30 m before passing through the operated gate valve. From the gate valve, the pressure drops almost 10 m, and the water discharges into the channel tank located downstream of the PVC pipe.

In the experimental tests, the 4 pressure transducers were installed along the entire pipeline, especially where transitory events are caused, i.e. in the PVC pipe. So two of them were installed in PVC pipe being one located immediately upstream of the closing valve, and the other transducer located 2 meters before the GI connection. The remaining two were located in the GI pipe, one near the connection of GI and PVC pipes, and the last one installed upstream of the system, next to the pump (Figure 2).

![Figure 2: Laboratory of Guanajuato’s University (scheme).](image)

These transducers were connected to an oscilloscope in order to measure the pressure as a function of time. The oscilloscope shows the passage of the acoustic wave past each pair of pressure transducers.

To perform surge experiments, a steady flow was created from the upstream tank through the GI pipe, using the pump, the inlet valve and the following manual control valves, up to the discharge tank. With the system in continuous operation, the manual valve installed in the PVC pipe was open and the manual valve closing to the discharge tank was closed, being the system constituted by the two mentioned pipes. Then a known discharge value from the inlet valve to the discharge channel was set, maintaining a steady flow condition. After creating the conditions for steady regime, a surge was created by quickly closing the butterfly valve. Instantly, the closing time of the valve was measured, using a stopwatch, and the oscilloscope started to record the pressure surge. Repeated tests were performed for each discharge values (see Table 1).

According to the type of support and material of the pipes, the celerity was also calculated through:

\[
c = \sqrt{\frac{K/\rho}{1 + \frac{K}{E_0} D e^{-\alpha}}} \tag{1}
\]

where \(\alpha\) is a constant that depends on the load conditions and type of wall, \(K\) the volume compressibility modulus, \(E_0\) is the elasticity modulus of the pipe material, \(D\) the internal diameter and \(e\) the thickness.

The celerity values of each pipe are presented in Table 2.
Table 1: Discharge values tested in the facility.

<table>
<thead>
<tr>
<th>Flow meter</th>
<th>Scenario 1</th>
<th>Scenario 2</th>
<th>Scenario 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ultrasonic</td>
<td>9.13</td>
<td>1.034</td>
<td>8.28</td>
</tr>
<tr>
<td>Conventional</td>
<td>9.35</td>
<td>1.085</td>
<td>8.50</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Scenario 1</th>
<th>Scenario 2</th>
<th>Scenario 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average</td>
<td>1.034</td>
<td>1.160</td>
<td>0.940</td>
</tr>
</tbody>
</table>

Table 2: Celerity values for each type of pipe and velocity values in each scenario.

<table>
<thead>
<tr>
<th>Material</th>
<th>$D_i$ (m)</th>
<th>$c$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PVC</td>
<td>0.1073</td>
<td>341.65</td>
</tr>
<tr>
<td>GI</td>
<td>0.10474</td>
<td>1310.12</td>
</tr>
</tbody>
</table>

Figure 3: Pressure diagrams obtained from the pressure transducers for each scenario.

For each scenario, pressure diagrams associated to the quickly closure of the butterfly valve, were obtained. Figure 3 shows the diagrams obtained from the 4 pressure transducers. Channel 1 (CH1) represent the upstream transducer, located in GI pipe, CH2 the next transducer, and so on, being CH4 the last transducer located near the butterfly valve. In each scenario, as soon as the butterfly valve is closed, CH4 immediately records the pressure variation, subsequently CH3 starts to record after 0.24 s (time the waves take to cross the pipe length until it reaches the next transducer), followed by CH2 and CH1. A relief wave travels downstream towards the butterfly valve and reaches it at a time $t = T = 2L/c$. A sudden change in pressure occur in CH4, leading to an overpressure after 0.56 s ($T = 2L/c$). This over
pressure coincides with the instant that the structure starts to move, immediately after the relief wave reaches the closing valve.

Figure 4: Example of an overpressure obtained in each scenario.

4 COMPUTATIONAL MODELS

After calibrating and studying the flow behaviour, the displacements of the pipes were studied with the use of a CFD model. First the 3D geometry model was introduced, where two types of domains were created, Fluid and Structure. For the fluid flow, the boundary conditions correspond to the tests made “in situ”, where upstream the constant pressure inlet obtained in CH1 was introduced and downstream a pressure outlet was set. The pressure outlet corresponds to maximum overpressure achieved in each scenario due to the rapid closing of the butterfly valve. Figure 5 represents the 3D model with the boundary conditions identified in the fluid domain.

Figure 5: 3D CFD model: (a) fluid domain; (b) structure domain.

In the structure domain, PVC material was set (Figure 5 b) and the supports of the laboratory facility were introduced, restricting its movements, according to its position.

Regarding the boundary conditions, the solution of fluid-structure interaction means that neither a fluid nor a structural system can be solved independently due to the unknown forces in the interface region [6]. Thus, the solution is based on relations of continuum mean mechanics, solved based on finite element methods. Hence, the results behind the transference of forces and momentums between the fluid and the pipe wall during the occurrence of the unsteady condition created by the rapidly close of the butterfly valve, in each scenario, are presented in Figure 6.
The maximum displacements obtained for each scenario, after the fluid simulation, are associated to the fluid stresses obtained for the last instant. Due to unbalanced interior pressure value in the butterfly valve, the inducing forces responsible for the system’s displacement is about 4024 kN, 4615 kN and 3597 kN, in scenario 1, 2, and 3, respectively.

5 CONCLUSIONS

The coupling fluid-structure phenomenon is seen in this ongoing research, as a great approach to describe a fluid-structure interaction case. The hydrodynamic pressure tends to be amplified due to the fluid motion induced by perturbations, which in turn induces some additional structure deformations. Consequently these deformations will also modify the hydrodynamic pressure. Through this work it will be possible to compare the results obtained in the 3D model to those obtained in “situ”.

REFERENCES