



Jul 1st, 12:00 AM

## 3D computational model of external intrusion in a pipe across defects

P. Amparo López-Jiménez

Jesus Mora-Rodriguez

Francisco José García-Mares

Vicente S. Fuertes-Miquel

Follow this and additional works at: <https://scholarsarchive.byu.edu/iemssconference>

López-Jiménez, P. Amparo; Mora-Rodriguez, Jesus; García-Mares, Francisco José; and Fuertes-Miquel, Vicente S., "3D computational model of external intrusion in a pipe across defects" (2010). *International Congress on Environmental Modelling and Software*. 520.

<https://scholarsarchive.byu.edu/iemssconference/2010/all/520>

This Event is brought to you for free and open access by the Civil and Environmental Engineering at BYU ScholarsArchive. It has been accepted for inclusion in International Congress on Environmental Modelling and Software by an authorized administrator of BYU ScholarsArchive. For more information, please contact [scholarsarchive@byu.edu](mailto:scholarsarchive@byu.edu), [ellen\\_amatangelo@byu.edu](mailto:ellen_amatangelo@byu.edu).

## 3D computational model of external intrusion in a pipe across defects

**P. Amparo López-Jiménez, Jesús Mora-Rodríguez,  
Francisco José García-Mares, Vicente S. Fuertes-Miquel**  
*Hydraulic and Environmental Engineering Department,  
Universidad Politécnica de Valencia, Valencia, Spain.*  
[palopez@gmmf.upv.es](mailto:palopez@gmmf.upv.es), [josmorod@doctor.upv.es](mailto:josmorod@doctor.upv.es), [fjgarcia@gmmf.upv.es](mailto:fjgarcia@gmmf.upv.es),  
[vfuentes@gmmf.upv.es](mailto:vfuentes@gmmf.upv.es)

**Abstract:** Water distribution networks are not completely physically safe and the possibility of entrance of external fluid inside the disinfected current of water exists, when some factors occur. This phenomenon is termed pathogen intrusion and can cause a pollution problem on the network. Polluted drinking water can generate waterborne disease outbreaks. Therefore pathogen intrusion is a phenomenon causing the degradation of drinking water quality. Three factors are needed: a pollution source, a way of entrance and a mechanism that produce the event. Furthermore, the presence of physical failures in the water network gives an approximation of the hydraulic performance; if the network is near to the optimal performance, this will have less possibility to generate an intrusion event, while the performance goes descending the possibility to present an intrusion grows. This work is focused on analyzing the relation of the ways of entrance (represented with failures in pipes) and the mechanism that generate the intrusion (negative pressure in the pipe). The flows of water are analyzed based on the pressure that generates the intrusion in the pipe by means of experimentation and three dimensional CFD modelling. The objective of this contribution is to identify by means of experimental models and with a numerical simulation of one of them, the potential relation among the intrusion flow and the pressure in the failure zone, to represent the event in a numerical way and to show the achievements of this model.

**Keywords:** *water quality model, pathogen intrusion, computational fluid mechanics*

### 1. INTRODUCTION.

#### 1.1 The problem of external intrusion into pipes

Water distribution systems conduct drinking water to supply the population. Water quality is controlled by the water treatment plant. However, the networks are not completely physically safe. Some operations by network staff can cause entry of external fluids, coming from the external environment surrounding fluids. This phenomenon is termed pathogen intrusion and can cause pollution problems for the network, López et al. [2006]. This paper describes the analysis of pathogen intrusion into water distribution systems through experimental and numerical modelling in order to study one of the phenomena that cause drinking water contamination. Polluted drinking water can generate waterborne disease outbreaks (*wbdo*), several countries maintain historical registers of these events and some revised studies have classified the different routes of intrusion.

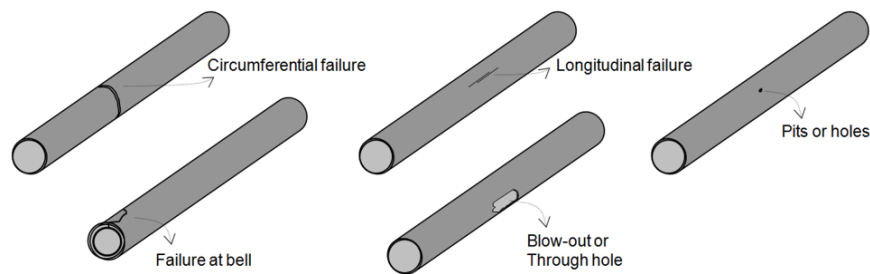
The objective of this paper is to make a steady state approach to the rate of flow intrusion occurring through an orifice representing the failure made by corrosion in a pipe. The theories of Torricelli and Favard are used to model water leakages and the performance of the distribution network.

## 1.2 Origin of intrusion into pipes

The report Pathogen Intrusion Into the Distribution System made by the American Water Works Association Research Foundation (AWWARF) and published in 2001 refers to cross connections as the direct entry pathways. These ways include: storage facilities, water main installation and repairs sites, and transitory contamination through water main leakage points Kirmeyer et al., [2001]. The present research is related to the last mentioned pathway: contamination through water main leakage points during negative pressure events.

During the last years the proportion of drinking water outbreaks reports varies on the origin place, Kirmeyer et al., [2001]. In 2006 a scheme was presented differentiating *wbdo* related with the water system deficiency. Sources were classified on five categories, Craun et al. [2006]: Deficiency in water treatment, distribution system deficiency, untreated ground water, miscellaneous or unknown deficiency and untreated surface water. The analysis was made during the period since 1971 to 2002 in the United States. The causes of this analysis are related with the evolution of the distribution system deficiency. Water distribution system deficiencies have increased due to *wbdo*. These deficiencies tend to be small, as contamination usually affects only a portion of the distribution system, limiting the potential exposure. Although a chemical etiology is often found (35% of the *wbdo*), distribution system *wbdo* are also caused by bacterial (17%), protozoan (14%), viral (4%), or undetermined (30%) pathogens. Leakages related to corrosion are failures usually producing hole shapes. Those failures are caused by electrochemical interaction resulted from the soil, water and degradation of the pipe material along the years. Metal Pipes are an important element on the water distribution networks, most of the half mains installed during the twenty century were metallic, Mora et al., [2001] and defects are so related to metallic materials.

Considering the kind of failure or the damage size, the corrosion could be classified on hole corrosion, or high grade or severity corrosion (Figure 1); and therefore the flow through this failures varies of different way. Analyzing some studies, when considering same pressure, the leakage flow vary with the pressure on exponential form, if the failure is a simple hole the flow is related with an exponent of 0.5 with the pressure value, but if the failure is due to extremely corrosion the relation between the operational pressure and the flow through the failure the exponent can arise up to 2.0 (Mora et al., 2008). Those results are considered in the case of leakage and the data was obtained from different references studies.



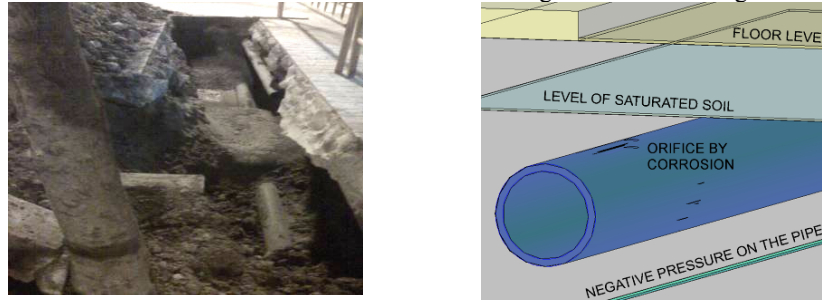
**Figure 1.** Defects in pipes caused by failures in operation

## 2. MODEL FOR EXTERNAL INTRUSION INTO PIPE ACROSS DEFECT

### 2.1 Experimental prototype

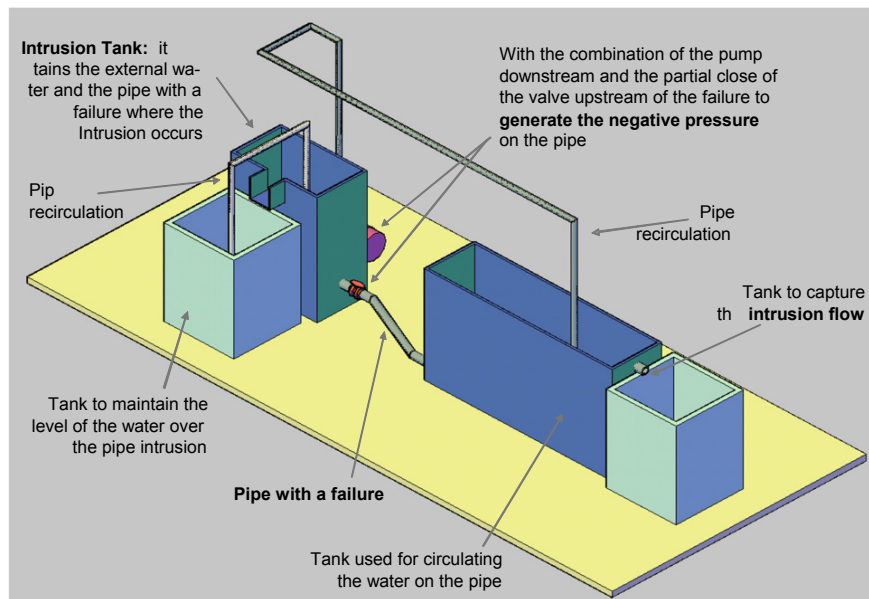
In the proposed model, the failure of the pipe was represented by a circular orifice; this geometry was selected in order to control the data on the simulations model. The pathogen intrusion is established by a negative pressure on the pipe. Contaminated water is represented by a source of constant level of water over the pipe. These conditions represent a typical leakage that is situated on a saturated environment with a hydraulic level of water (Figure 2). The size of the orifice is determined according to the theory of small orifices

discharges. There are two different type of small discharge according to the relation between the size and thickness of the orifice. The discharge could be thin or gross wall.



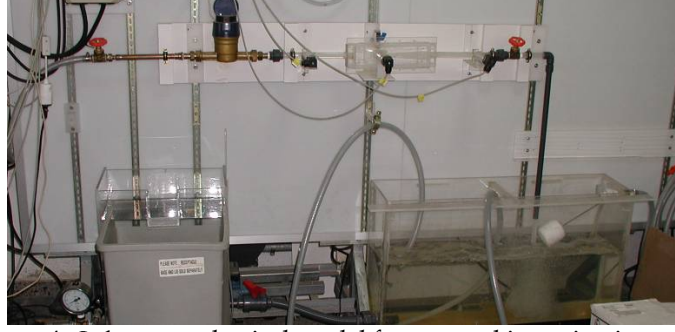
**Figure 2.** Physical problem analysed by prototype.

In this paper we present the study of the behaviour of three orifices that were simulated experimentally: the first diameter of 1.5mm in a wall thickness of 2.7mm, the second one a diameter of 4.0mm in a thickness of 2.7mm and finally, a diameter of 5mm diameter in a 2.1mm wall thickness, the scheme of physical model is show in figure 3. Finally we represented the orifice of 1.5mm diameter in a 3D numerical model.



**Figure 3.** Schematic distribution for experimental device developed for modelling external intrusion into pipe.

The prototype was built in laboratory representing the leakage of the pipe with an orifice. The first component to be considered is a suitable assembly developed in the laboratory (Figure 4). It is oriented to obtain the hydrodynamic parameters, both velocity and pressure around the orifice in order to get the physical data occurring during intrusion.



**Figure 4.** Laboratory physical model for external intrusion into pipe.

Negative pressure has been simulated with a pumping system in collaboration with a partial close of valve. The experiment has been designed to measure pressure and velocity in certain points. A suitable interface has been designed by means of Labview. The intrusion flow was taken in a volumetric way; in each one of the simulations a specific time was established in order to obtain this intrusion flow. In this physical model each scenario was simulated five times for every one of the five different opening of the main valve. Considering the theory of flow through orifices during the simulations was obtained the results depicted in Table 1.

**Table 1.** Experimental simulations.

Parameter	Orifice 1	Orifice 2	Orifice 3
diameter (d) mm	1.5	4	5
thickness (t) mm	2.7	2.7	2.1
kind of wall (w = t/d)	1.8	0.68	0.42
the operation	Submerged		
differential head ( $\Delta h$ ) mwc	2.3 to 5.7	1.0 to 3.4	0.4 to 7.2
relative dimension ( $d/\Delta h$ )	$6.5 \times 10^{-4}$ to $2.6 \times 10^{-4}$	$4.0 \times 10^{-3}$ to $1.2 \times 10^{-3}$	$1.3 \times 10^{-2}$ to $6.9 \times 10^{-4}$
velocity (v) m/s	5.4 to 8.3	3.6 to 6.1	2.4 to 8.3
Reynolds number	$8.1 \times 10^{-3}$ to $1.3 \times 10^{-4}$	$1.4 \times 10^{-4}$ to $2.5 \times 10^{-4}$	$1.2 \times 10^{-4}$ to $4.2 \times 10^{-4}$
Discharge Coefficient	0.64 to 0.63	0.63 to 0.62	0.63 to 0.61

### 2.3 CFD model for intrusion in pipe

The equations that describe the movement of a fluid are obtained by application of the continuity and angular momentum equations (Navier-Stokes equations). These equations are:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \quad (1)$$

$$\rho \frac{\partial \vec{V}}{\partial t} + \rho (\vec{V} \cdot \nabla) \vec{V} = -\nabla p + \rho \vec{g} + \nabla \cdot \tau_{ij} \quad (2)$$

where  $\rho$  is the density of the fluid,  $V$  the velocity field,  $p$  the pressures field,  $g$  the forces by volume unit which the fluid is put under, and  $\tau_{ij}$  is the voltage tensor.

In the CFD analysis, one of the basic aspects that are necessary to establish is the turbulence model. The more used model for air and water flows with highly turbulent regimes is the model  $k$ - $\epsilon$  proposed by Launder and Spalding [1972]. In this model, the turbulent kinetic energy  $k$ , and their ratio of dissipation  $\epsilon$  are obtained from the following equations.

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (3)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (4)$$

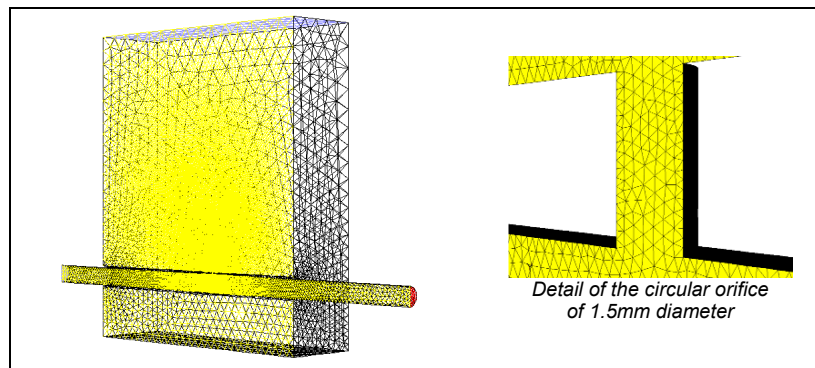
In these equations,  $G_k$  represents the generation of turbulent kinetic energy originated by the variations of the components of the average velocity of the flow;  $G_b$  represents the kinetic energy generated by boundary push;  $Y_M$  is the contribution of the pulsatile expansion associated to the compressible turbulence;  $C_{1\varepsilon}$ ,  $C_{2\varepsilon}$  y  $C_{3\varepsilon}$  are constants;  $\sigma_k$  y  $\sigma_\varepsilon$  are the Prandtl numbers for  $k$  and  $\varepsilon$  respectively; and  $S_k$  y  $S_\varepsilon$  respectively represent a global variation in the time of parameters  $k$  and  $\varepsilon$  that can be defined of independent form of the rest of variables. On the other hand, turbulent viscosity  $\mu_t$  calculates by means of a combination of the values of  $k$  and  $\varepsilon$ . The constants  $C_{1\varepsilon}$ ,  $C_{2\varepsilon}$ ,  $C_{3\varepsilon}$ ,  $\sigma_k$  and  $\sigma_\varepsilon$  adjust from the obtained results of experimental form. However, the values contributed by Launder and Spalding [1972] have been very effective with turbulent air flows and turbulent water flows.

In order to be able to solve the previous system of equations it is necessary to turn the system (3) and (4) into algebraic expressions by means of a discretization process. This discretization implies dividing the system in smaller volume, solving the problem in each element of a smaller mesh.

Thus, our numerical model simulates the prototype, using a program based on CFD, which displays the fields of physical components. In order to simulate numerically the five scenarios, the software ©FluentInc has been used. Focused on the conditions model, the RNG  $k$ - $\varepsilon$  turbulent model is used. Equations consider some terms that improve the accuracy for flows that are rapidly filtered, solve accurately the phenomenon of vortices in the flow, and also manage with differential effective viscosity to consider effects of small Reynolds numbers [Fluent, 2005]. Some of the characteristics of the models are the following ones: first-order discretization was used for numerical method. The gradient option was modified from cell-based to node-based in order to optimize energy conservation; this option is more suitable for tri-elements face meshes, [Fluent, 2005]. The mesh was generated with almost 130,000 nodes, 1,250,000 faces and almost 600,000 cells. Finally, the computation made raised up to 2500 iterations to achieve the best convergence conditions. Table 2 and Figure 5 show the incoming data for simulations and boundary conditions used to model the geometry and the case.

**Table 2.** Initial numerical data simulations.

Case	$V_{in}$ (m/s)	$P_{out}$ (mwc)	$P_{sla}$ (mwc)	Re	TI (%)	Dh (m)
1	0.979	-5.35	0.36	$2.65 \times 10^4$	4.48%	$2.72 \times 10^{-2}$
2	0.998	-4.28		$2.70 \times 10^4$	4.47%	
3	0.999	-3.12		$2.70 \times 10^4$	4.47%	
4	1.023	-2,51		$2.76 \times 10^4$	4.46%	
5	1.039	-1.96		$2.81 \times 10^4$	4.45%	



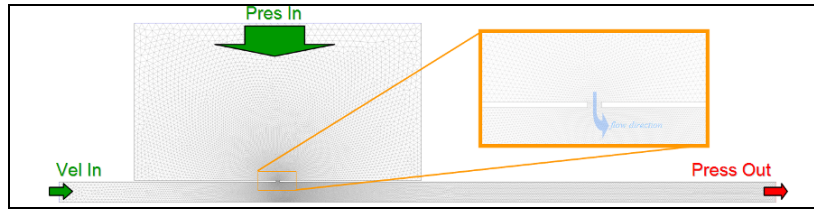


Figure 5. Boundary conditions and mesh details for the computational model.

#### 4. RESULTS

By both, mathematical and physical models, correlations are proposed to have a better knowledge of quantities that can not be measured during a flow intrusion through an orifice. By CFD models we can study aspects of turbulence, pressure fields, concentrations, etc. existing in mixing processes related to external intrusion.

The intrusion model was similar to the Torricelli's Theorem. In general, the drop of pressure and flow in orifices present a relation between both magnitudes. More modern expressions for this relation in leakages was established by May [1994]. He proposed the Favad theory: the flow depends directly on the pressure with a variable exponent, eq. (6). The exponent varies from 0.5 to 2.5; according to the expansion of the orifice or failure. For some individual leakage path, also the failure section depends of pressure. The analogy of intrusion flow with both theories, Torricelli and Favad is considered.

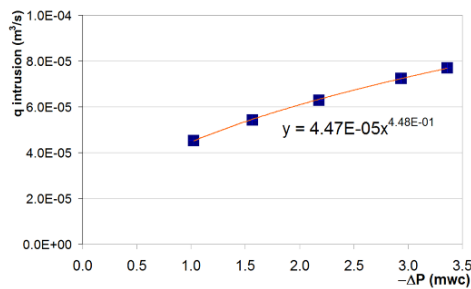
$$Q = P^N \quad 0.5 < N < 2.5 \quad (6)$$

Where Q is the flow that leaks trough an orifice, P is the pressure on the section of the orifice and N is the exponent where can take values between 0.5 and 2.5 depends on the type of expansion of the failure. In the case of the two larger diameters, the results of the relation between the pressure on the section failure and the intrusion flow are presented on the Table 3, including the discharge coefficient that is obtained for every case.

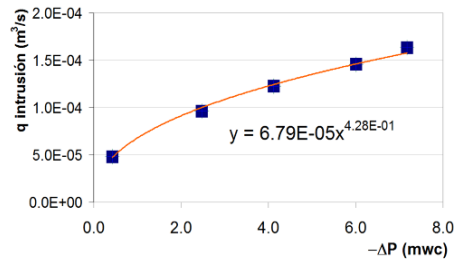
Table 3. Experimental Results

Interior Diameter pipe (mm)	Roughness (mm) data from catalogue	Circular Failure		Potential Equation $\Delta P$ (mwc) v.s. $Q_{intr}$ (m <sup>3</sup> /s)	
		$d_f$ (mm)	$A_f$ (m <sup>2</sup> )	Exponent N	$C_d$ ( $Y = C_d X^N$ )
27.2	0.0025	4.0	12.57	0.45	4.47E-6
28.2	0.0025	5.0	19.64	0.43	6.79E-6

On these results, it can see that the exponent is near to 0.5; but in both cases this value is underestimated. On the first case with a diameter of 4mm and the range of the pressure on the experiments is from 1.0 to 3.4mwc the exponent is 10.4% less than the theoretical value. On the case of the orifice of 5mm diameter, this was experiment on a range from 0.4 to 7.2mwc and the difference with the theoretical value was of 14.5%. On Figure 6, we can observe that near to the origin the curvature of the equation presents a more abrupt slope.



Results from 4mm diameter experiment



Results from 5mm diameter experiment



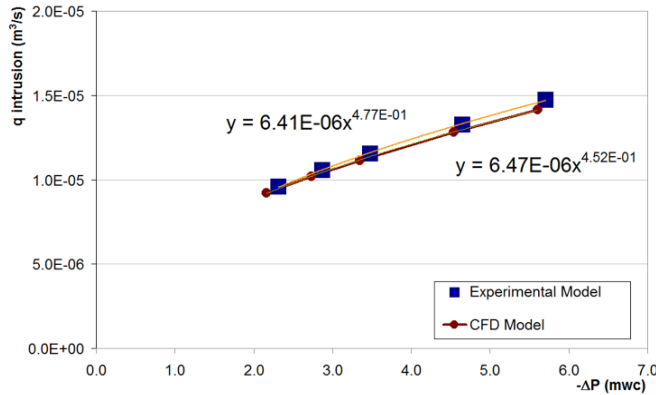
**Figure 6.** Experimental Results according to the relation between pressure and velocity.

On the case of the small diameter (1.5mm), besides the experimental model, was doing a numerical model, the summary of the models is presented on the table 4. In the experimental one, the exponent was near to the theoretical value; in this case the 0.48 presents a difference of 4.7%. On the numerical results the exponent underestimates so much the experimental as the theoretical models; in general the hydraulics CFD models underestimate the physical data. In this case the numerical result differs 5.1% from the experimental one.

**Table 4.** Numerical Results

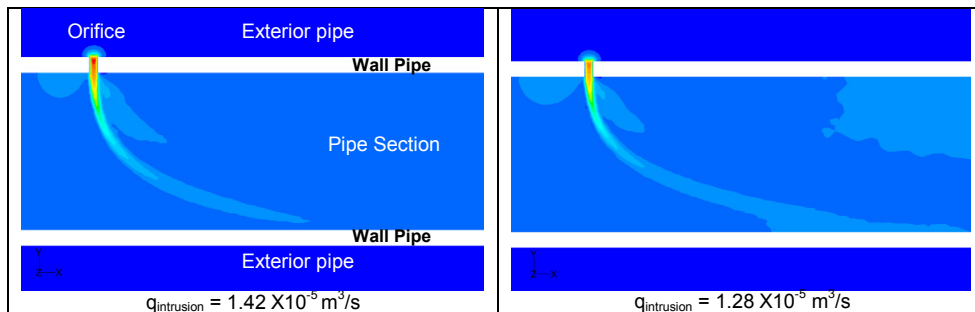
Interior Diameter pipe (mm)	Roughness (mm) data from catalogue	Circular Failure		Potential Equation $\Delta P$ (mwc) vs $Q_{intr}$ (m <sup>3</sup> /s)	
		$d_f$ (mm)	$A_f$ (m <sup>2</sup> )	Exponent N	$C_d$ ( $Y = C_d X^N$ )
27.2	0.0025	1.5	1.77	0.48	6.41E-6
27.2	0.0025	1.5	1.77	0.45	6.47E-6

On Figure 7, the equation can be observed, considering both experimental and numerical values. In all the models the adjustment of the data is more than 0.99 of R<sup>2</sup>. In this case the experimental values were on a range from 2.3 to 5.7mwc.

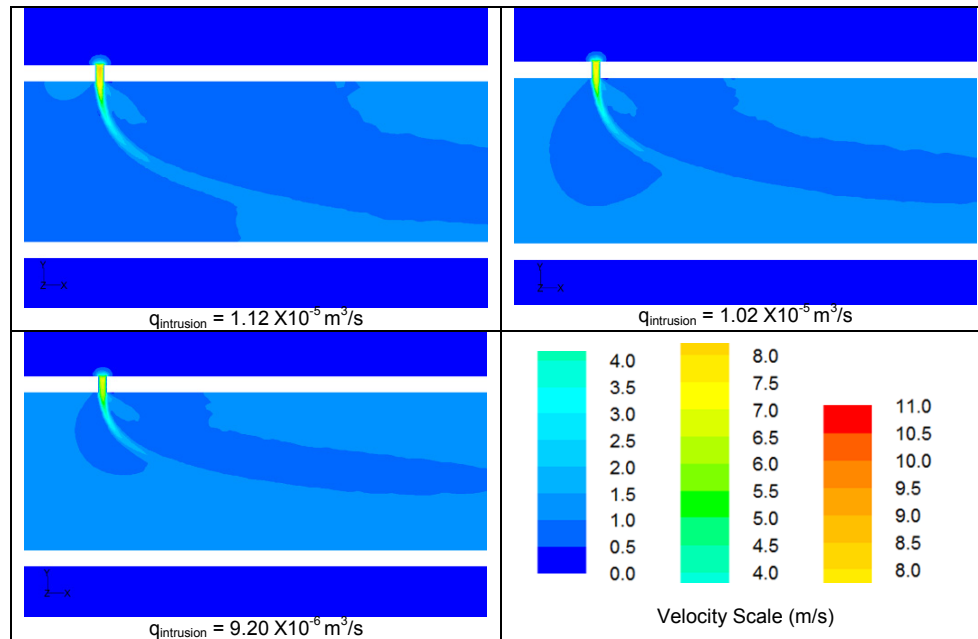


**Figure 7.** Experimental and Numerical Results of the orifice of 1.5mm diameter.

Finally, the numerical model obtained an exponent value of 0.45; this value underestimates 9.5% the theoretical one of 0.50. The configuration of the velocity on the zone of the orifice is presented by the following figure 8.







**Figure 8.** Section velocity configuration on the intrusion zone.

## 5. CONCLUSIONS

In this contribution hydraulics factors contributing to generate the pathogen intrusion were presented. It was obtained the mathematical correlation between the pressure and the flow entering through an orifices simulated during events of negative pressure.

Three experimental simulations were performed in different range of pressure; in the case that the experiment value of pressure is near to zero, the exponent of the relation of flow and pressure on the intrusion presents more difference with its theoretical value. According to the experimental models, this type of orifices simulated do not expand near the defect, as Favard theory depicts.

Theoretical comparisons Torricelli's and Favard theories have been used, obtaining an exponent from the models near to 0.50; agreeing very much with theoretical predictions of this value. On the three experimental cases, the exponent value was less than the theoretical value.

The numerical model appropriately represents the experimental simulation, demonstrating that this kind of models could generate some special cases that in physical way are complex to evaluate. On those cases authors are working in order to obtain the physical behaviour of the intrusion combining with the consideration of the external porous media surrounding the orifice pipe and the cases with pressure transient simulations on the pipe. These experimental results will be used to describe numerically a wider range of scenarios and to obtain a quantify of flow entering during a pathogen intrusion with more physical elements.

## ACKNOWLEDGEMENTS

This work has been possible inside the actions developed by the researchers of UPV involved in the project DANAIDES: Desarrollo de herramientas de simulación para la caracterización hidráulica de redes de abastecimiento a través de indicadores de calidad del agua. REF. DPI2007-63424. Ministerio de Educación y Ciencia de España.

## REFERENCES

- Craun, F. M., Craun, F. G., Calderon, L. R., Beach, J. M., Waterborne outbreaks reported in the United States, *Journal of Water and Health*, 04 (Supl. 2), 9-30, 2006.
- Fluent, User's guide, Fluent 6.2, © Fluent Inc., 2005.
- Kirmeyer G. J., Friedman M., Martel K., Howie D., LeChevalier, M., Abbaszadegan M., Karim M., Funk J., Harbour J., *Pathogen intrusion into distribution systems*, AWWARF and USEPA, USA., 2001.
- Launder, B. E. and Spalding, D. B., *Mathematical models of turbulence*, Academic Press Inc., 169pp. London, 1972.
- López, A., Martínez, J., López, G., Fuertes, V., Análisis de la problemática de calidad de los abastecimientos: desde la planta de tratamiento hasta el grifo, V Congreso ibérico sobre gestión y planificación del agua, Faro, Portugal, Dec, 4th, 2006.
- May, J., Pressure-dependent leakage, *World Water and Environmental Engineering.*, October, 1994.
- Mora, J., López, A., Delgado X., Alonso, C., Estudio sobre la modelación de defectos en tuberías, *VIII Seminario iberoamericano, alterações climáticas e gestão da água e energia em sistemas de abastecimento e drenagem (SERA)*, Lisboa, Portugal, July, 2008.