

NUMERICAL MODELING OF EFFLUENT DISPERSION IN RIVERS WITH DIFFERENT LONGITUDINAL DIFFUSION COEFFICIENTS

A.G.S.L. Ritta, almirslritta@gmail.com

P.S. Estolano, pestolano22@hotmail.com

V.A.M. Martins, vinicius_martins@id.uff.br

R.M. Moreira, rmmoreira@id.uff.br

Fluminense Federal University, R. Passos da Pátria, 156, bl.D, sl.563A, Niterói, R.J., 24210-240, Brazil.

Abstract. *The study of pollutant dispersion is very important to assess the environmental impact of new industrial developments in rivers and to develop a mitigation plan in case of any toxic product spill. The purpose of this paper is to study numerically the effects of longitudinal diffusion models in the dispersion of effluents in rivers and channels. Continuity and Navier-Stokes equations are solved via a finite volume method with a homogeneous multiphase condition for a 2D channel. The velocity field is then used to solve the mass transport of a soluble effluent in the time domain considering different longitudinal diffusion coefficients. Results are shown in terms of the concentration field along time and space, showing significant differences of peak concentration and time of arrival of the plume depending on the diffusion model employed.*

Keywords: *River dynamics, longitudinal dispersion coefficient, computational fluid dynamics,*

1. INTRODUCTION

Water resource always has been important to humanity. Since ancient times, civilization has been developing its society near places that offers access to water sources. Nowadays, water consumption has been gradually increasing and because of this increase of demand it has implications to humanity. Hence, researches are now finding ways and methods to preserve or minimize the implication of excessive water use that has already been caused by humans. The knowledge of hydrodynamics and dispersion in rivers is the key to overcome future endeavours related to river pollution (Dimian *et al.*, 2013).

The existence of computers with high processing capacity and the advances in numerical analysis had opened new horizons in flow modelling for rivers and channels. The development of new numerical techniques applied to different flow regimes made possible to simulate some phenomenas in other ways that are not only experimental. Numerical simulations require validation through analytical solutions, *in situ* measurements or laboratory experiments. In general, analytical methods are applicable only to simple boundary value problems. However, they provide important information on the role of significant parameters that can be considered in modelling. *In situ* measurements are more realistically to represent the hydrodynamics and the spreading of effluents into bodies' receptors (Modenesi *et al.*, 2004). On the other hand, they can be high cost and often can not be done for security reasons, difficulties to access the area or non-authorization from environmental agencies.

The performance of reduced scale experiments, applying the concepts of dimensional analysis and dynamic similarity to real physical problem, is an alternative to check the hydrodynamic and the models dispersion used in numerical models. Despite the difficulty to reproduce with accuracy field conditions, in the laboratory is easier to keep controlled the boundary conditions. Thus, mathematical model can represent more realistically the problem, which is being studied.

In this paper, the hydrodynamic and the kinetic of effluent discharge in a reduced model are simulated through COMSOL Multiphysics which is a finite element analysis, solver and simulation software/package for various physics and engineering applications, especially coupled phenomena, or multiphysics. The measurement of concentration and arrival times in downstream region of the discharge are analysed and compared with the classical model of linear transport for instantaneous and continuous discharge.

2. LINEAR TRANSPORT MODEL

According to Tucci (1998) the one-dimensional, linear transport and semi-empirical model for concentration of a given effluent at time t and space x , obeys the following transport equation:

$$\frac{\partial C}{\partial t} + v \frac{\partial C}{\partial x} = E \frac{\partial^2 C}{\partial x^2} - k_1 C \quad (1)$$

where $C(x,t)$, v , E , k_1 , respectively, represent the function concentration, the speed of river, longitudinal diffusion coefficient and decay effluent coefficient.

For a continuous pollutant load, the differential Eq. (1) has the following analytical solution,

$$C(x, t) = \frac{C_0 e^{-\left(\frac{k_1 x}{v}\right)}}{2} \left\{ \operatorname{erf} \left[\frac{x - v(t - t_0)(1 + n)}{\sqrt{4E(t - t_0)}} \right] - \operatorname{erf} \left[\frac{x - vt(1 + n)}{\sqrt{4Et}} \right] \right\} \quad (2)$$

where C_0 is the initial concentration considering a homogeneous mixing of the pollutant load and n is given by,

$$n = \frac{k_1 E}{v^2} \quad (3)$$

According to Devens *et al.* (2006) for an instantaneous discharge, with initial concentration C_0 , the differential Eq. (1) has the following analytical solution,

$$C(x, t) = \frac{C_0}{\sqrt{4\pi Et}} e^{-\left(\frac{(x-vt)^2}{4Et} - k_1 t\right)} \quad (4)$$

To examine the effects of longitudinal coefficient (E), two analytical analyses were calculated with different values of E . In Figure 1, there is a strong decay of maximum concentration over time. It should also be noted the role of the diffusion coefficient, amplifying, for example, within 2 km of the plume spreading after 10 hours of the release was made.

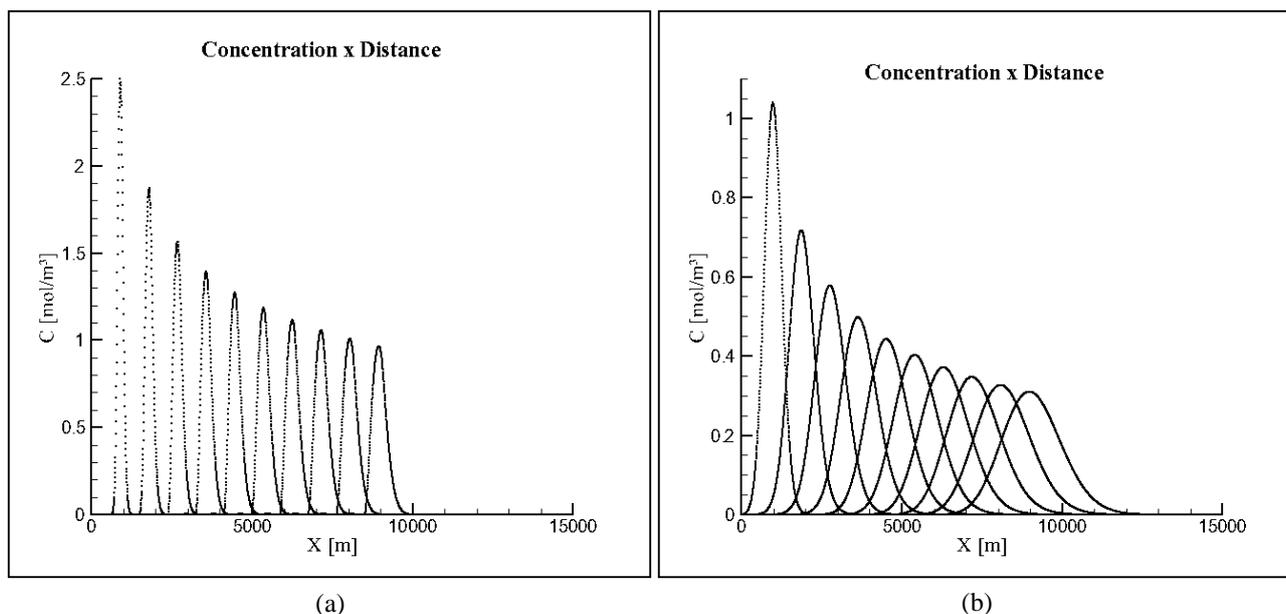


Figure 1: Concentration profiles for instant loading pollutant over 10 hours ($v=0.25$ m/s). (a) $E=0.021$ km²/day (b) $E=1.0$ km²/day.

3. COMPARISON BETWEEN ANALYTICAL AND NUMERICAL METHODS

The non-linear transport equation for the concentration of a given effluent is given by Eq. (5),

$$\frac{\partial C}{\partial t} + \vec{u} \cdot \nabla C = E \nabla^2 C - k_1 C \quad (5)$$

where \vec{u} is the velocity field.

For determining \vec{u} , the mass conservation Eq. (6) and amount of movement Eq. (7) are solved via finite element method,

$$\nabla \cdot \vec{u} = 0 \quad (6)$$

$$\rho \frac{D\vec{u}}{Dt} = -\nabla P + \mu \nabla^2 \vec{u} \quad (7)$$

Where ρ and μ are, respectively, the density and the dynamic viscosity of water and P is the modified pressure. The turbulence model $k-\varepsilon$ was applied in simulations.

For the instantaneous discharge scenario in the channel, it was considered a distance of 10km and release with initial concentration of 1mol/L. In order to simulate the instantaneous scenario, the Eq. (8) was used to simulate:

$$C = C_0 e^{-\left(\frac{t}{\tau}\right)^{2.1}} \quad (8)$$

Using the above equation and the initial and boundary conditions, Fig. 2 was plotted.

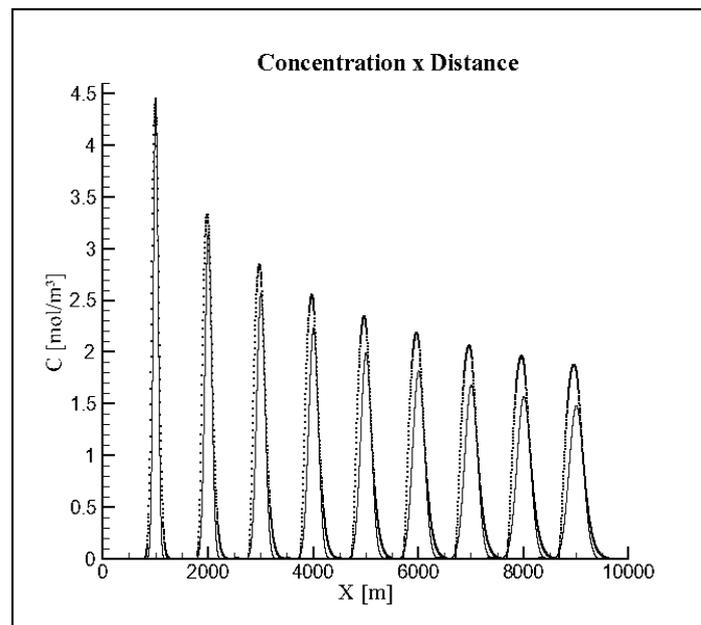


Figure 2 - Analytical profile concentration (black-line) and numeric profile concentration (black-dots) for an instantaneous load pollutant over 10.000 seconds using $v = 1.0\text{m/s}$ and $E = 1.0 \text{ km}^2/\text{day}$.

For a continuous discharge scenario, it was considered a distance of 10km and with standard concentration of 1mol/L. Using these initial and boundary conditions, Fig. 3 was plotted:

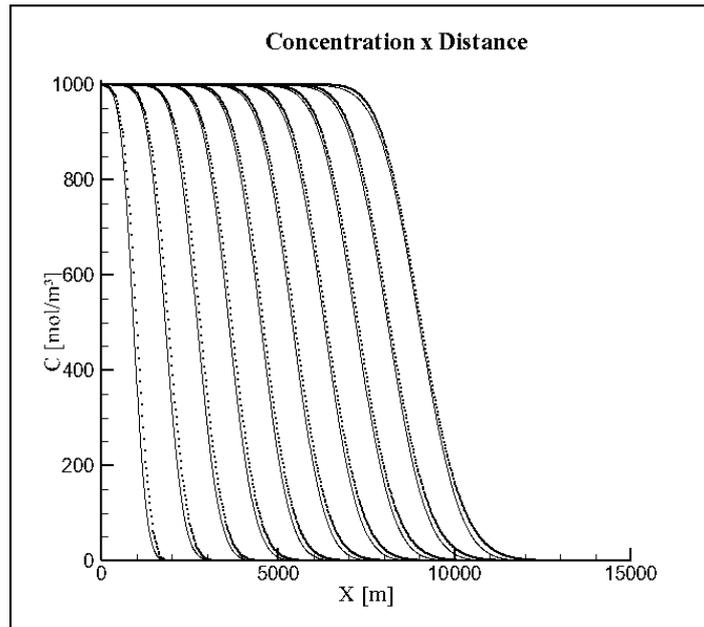


Figure 3 - Analytical profile concentration (black-line) and numeric profile concentration (black-dots) a continuous load pollutant over 10.000 seconds ($v = 1.0\text{m / s}$). $E = 1.0 \text{ km}^2/\text{day}$.

Figure 2 shows the pollutant behaviour through time and distance. In the course of time, the peaks simulated by numeric method are decreasing more than the peaks calculated by analytical method. However, both curves keep the same behaviour and the centre of the peaks is following a pattern to most of them. On the other hand, Fig 3 has the same pattern of decay for both curves.

4. 2D ANALISYS

The study of effects of longitudinal diffusion coefficient was made to figure out the behaviour of the plume formed in a channel with 6m of length and 15cm of thickness. Figures 4, 5 and 6 are showing different plumes caused by the same inlet parameters, excepted by E , which is different for each figure. The coefficients used in these analyses were two constant coefficients (0.021 and $1.0 \text{ km}^2/\text{day}$) and a third value that is function of the turbulent flow μ_t (km^2/day). The inlet concentration, viscosity and density are, respectively, 1mol/L , $1 \times 10^{-3}\text{Pa.s}$ and 1000Kg/m^3 .

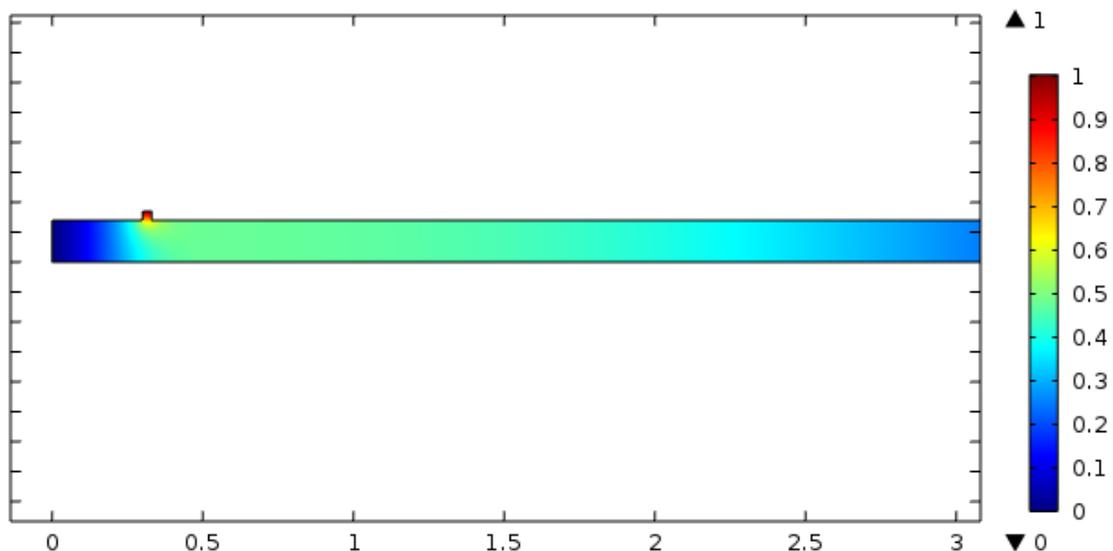


Figure 4 - Numerical analyse with $v=1\text{m/s}$ and $E= 0.021 \text{ km}^2/\text{day}$.

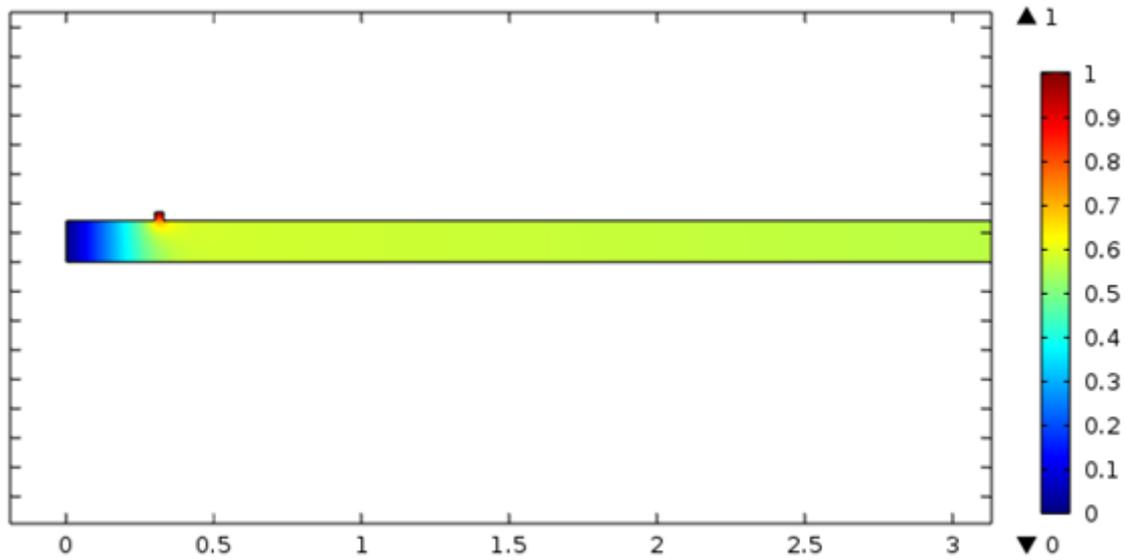


Figure 5 - Numerical analyse with $v=1\text{m/s}$ and $E= 1.0 \text{ km}^2/\text{day}$.

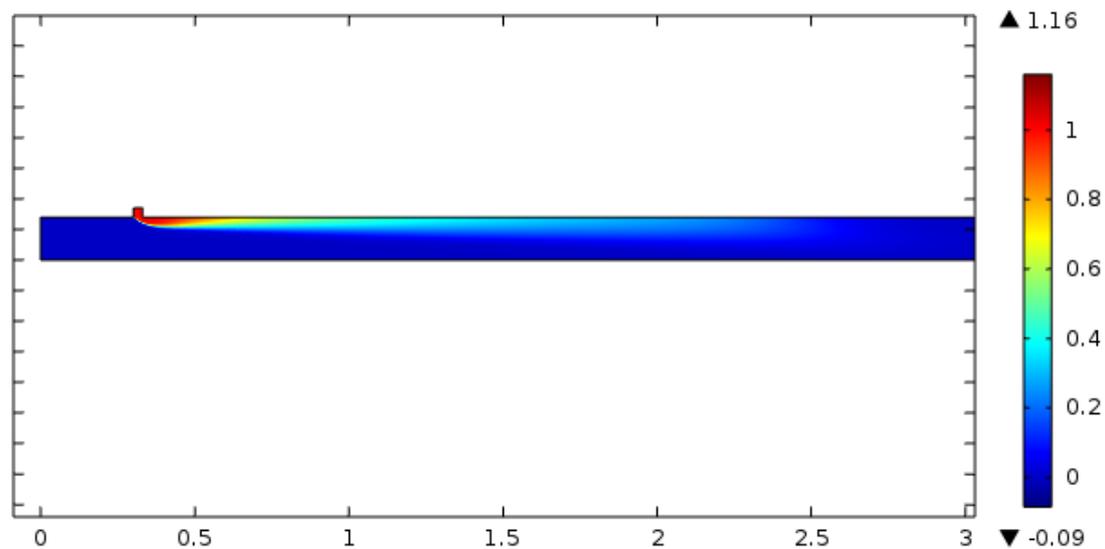


Figure 6 – Numerical analyse with $v=1\text{m/s}$ and $E= \mu_t$.

5. CONCLUSION

Owing to its theoretical interests and practical importance, the pattern of flow and mass transport process in channel has attracted much attention of researchers and engineers. In this paper, a 2D non-linear k- ϵ turbulence hydrodynamic and mass transport model are established and applied to simulate the flow and mass transport process in channel. The simulated results have been compared with the analytical data, and the agreement between both is generally good. The similarities in results in both studies (analytical and numerical) can prove that the numeric methods can be used to represent the real channel and consequently it can be used to examine the effects of longitudinal diffusion coefficient. Which showed an expressive difference between the plumes formed using different values of E. This study demonstrates that the 2D non-linear k- ϵ turbulence model can be used for analysing flow structures and pollutant transport in canal, this model is also can be applied to real river.

6. REFERENCES

Devens, J.A., Barbosa Jr., A.R. and Silva, G.Q., 2006. Modelo de quantificação do coeficiente de dispersão longitudinal de pequenos cursos de águas naturais. *Eng. Sanitária Ambiental*, Vol. 11, No. 3, pp. 269-276.

Dimian, M.F.A., Wadi, A.S. and Ibrahim, F.N., 2013. "The effect of added pollutant along a river on the pollutant concentration described by one –dimensional advection diffusion equation". *International Journal of Engineering Science and Technology (IJEST)*, Vol. 5, No. 9, pp. 1662-1671.

Modenesi, K., Furlan, L.T., Tomaz, E., Guirardello, R. and Núñez, J.R., 2004. A CFD model for pollutant dispersion in rivers. *Brazilian Journal of Chemical Engineering*, Vol. 21, No. 4, pp. 557-568.

Sperling, M.A., 2007. *Estudos e modelagem da qualidade da água de rios*. Editora UFMG.

Tucci, C.E.M., 1998. *Modelos hidrológicos*. Editora UFRGS.

7. RESPONSIBILITY NOTICE

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