PRE-PRINT (July, 2006)

Good mixing length: Digital simulation of fluid mixing with and without obstacles

Roberto Suárez Ántola[(1)(2)] rsuarez@ucu.edu.uy
Manuel R. Burgos[(1)] mburgos@ucu.edu.uy

[(1)] Departamento de Ingeniería Eléctrica, Facultad de Ingeniería y Tecnologías, Universidad Católica del Uruguay.
[(2)] Dirección Nacional de Energía y Tecnología Nuclear, Ministerio de Industria, Energía y Minería, Uruguay.

Abstract

The *good mixing length* of a tracer assures that the samples or measures taken are fair. A non homogeneous tracer mixing through the cross section of the fluid medium involved in the experiment (eg. a river or a pipe) may conduct to erroneous conclusions.

For establishing that length, a digital simulation of a two dimensional fluid flow, using Navier-Stokes equations, was done. A continuous tracer injection was simulated.

The good mixing length was studied in two cases, first with a free of obstacles situation and then the effect of a significant obstacle located after the tracer injection point.

As usual in practice, the good mixing length was estimated using a suitable upper bound for the concentration deviations from the mean in a given cross section.

An analytical discussion of the obtained results is done.

Introduction:

Classical tracer techniques applied to quantify industrial, environmental and hydrological processes rely upon the attainment of a good mixing of the tracer in a cross section of the flow, at the point of measurement. Let us suppose that a mass of tracer is suddenly injected in the instant $t = 0$ at a point P_0 of a cross section Σ_0 of a certain flow. Then if $c(t, P; 0, P_0)$ is the tracer concentration at the point P and the instant t in a cross section Σ located downwards in relation with the flow, it is possible to estimate the integral $I_c(P, P_0) = \int_0^\infty c(t', P; 0, P_0)dt'$. When this integral is independent of both points (the injection point and the measurement point) the good mixing condition has been attained. The **good mixing condition**, by definition is

attained when the following happens: the probability that a **particle of tracer**, that in $t = 0$ was located in $P_0 \in \Sigma_0$, crosses the element of area $\Delta\Sigma$ (that belongs to the cross section Σ) in some instant of time between 0 and t_0 is equal to the probability that an **inert particle**, initially located in some point $P'_0 \in \Sigma_0$, crosses the same element $\Delta\Sigma$ in some instant between 0 and t_0, with t_0 big enough. This is equivalent to the constancy of the integral I_c, as will be discussed below.

In case of a continuous injection of tracer in a stationary flow and with constant injection rate, the good mixing condition is attained when the concentration $c(t, P; 0, P_0)$ doesn't vary across a cross section of the flow, from a certain cross-section onwards.

For a model of a rectangular channel of constant cross section and infinite length, with a constant velocity flow, it is fairly easy to solve (by separation of variables or any other analytical technique) the three-dimensional advection dispersion equation for a tracer, both for a localized and sudden injection and for a localized (at the same point) continuous injection. Then the equivalence of the two mentioned criteria can be derived using as a starting point the analytical solutions. However, the exact equality between the constant value of I_c in different points for a sudden injection and the constant value of the concentration in different points of the same cross section for a continuous injection, is verified only asymptotically when the distance from the injection point tends to ∞ . Therefore, in order to work with finite distances a certain margin of error must be allowed.

Also in practice the good mixing condition is fulfilled only approximately, so that an upper bound to allowable deviations between the integrals or between the concentrations in a given cross section of the flow must be established both for field experiments and for the results of digital simulations.

The distance between the injection point and the nearest cross section that verifies the good mixing condition between the established bounds for the error, suitably measured along a representative flow line, is by definition the mixing length.

In this paper a digital simulation of the mixing process of a tracer in a two dimensional flow is used to estimate the mixing length. Also, an analytical discussion for a hydraulically closed system of the same type considered in the digital simulation is briefly done in the last part of the work.

Numerical simulation: the continuous model and its discretisation.

A laminar flow of a viscous, incompressible fluid is considered. A fluid in two dimension domain, $\Omega \subset \Re^2$ throughout time $t \in [0, t_{end}]$ is characterized by (as in GRIEBEL, M et al. 1997)

$\vec{u} : \Omega \times [0, t_{end}] \to \Re^2$ velocity field.

$p : \Omega \times [0, t_{end}] \to \Re$ pressure field.

$\varphi : \Omega \times [0, t_{end}] \to \Re$ density field.

As the fluid under study is incompressible, density changes are negligible, so we assume $\varphi(\vec{x}, t) = \varphi_\infty = const.$ Flow is described by a system of partial differential equations whose dimensionless form is given by

$$\frac{\partial}{\partial t}\vec{u} + (\vec{u}.\text{grad})\vec{u} + \text{grad}\, p = \frac{1}{\text{Re}}\Delta\vec{u} + \vec{g} \qquad \text{(momentum equation)}$$

$$\text{div}\,\vec{u} = 0 \qquad \text{(continuity equation).}$$

Where Re is the dimensionless Reynolds number of the flow (not the fluid) a relation between inertia and viscous forces, and $\vec{g}$ is the gravity force.

The components of the systems are the Navier Stokes equations.

For the two dimensions domain, if we introduce the scalar components of the corresponding vectors, $\vec{x} = (x, y)^T$, $\vec{u} = (u, v)^T$ and $\vec{g} = (g_x, g_y)^T$ the system may be written as:

$$\frac{\partial u}{\partial t} + \frac{\partial p}{\partial x} = \frac{1}{\text{Re}}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) - \frac{\partial (u^2)}{\partial x} - \frac{\partial (uv)}{\partial y} + g_x$$

$$\frac{\partial v}{\partial t} + \frac{\partial p}{\partial y} = \frac{1}{\text{Re}}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) - \frac{\partial (uv)}{\partial x} - \frac{\partial (v^2)}{\partial y} + g_y \qquad \text{(momentum equations)}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad \text{(continuity equation)}$$

At $t = 0$, initial conditions as $u = u_0(x, y)$ and $v = v_0(x, y)$ which satisfy the continuity equation were imposed.

As the discretisation is made on a rectangular regular matrix, velocity components normal and tangential to the boundaries, along cells limiting segments (ω_n and ω_t respectively) may be established. Velocities vanish in the boundary in a no-slip condition, i.e. no fluid penetrates the boundary and the fluid is at rest there. This is the case of the simulation carried out in this application.

A staggered grid was used for representing the domain, pressure and velocities components are imposed in different points of each cell, enabling the averaging of values between adjacent cells.

The developed software may simulate any input: impulsive, continuous or an arbitrary one, by presenting the proper value at the pipe input at each simulation step. The input may be applied at any portion of the system under study.

For each step, the velocity field is updated, obtaining a time evolving velocity field. With this field, a dispersion model is applied.

For studying the mixing characteristics, a continuous tracer injection was imposed at the input and the deviation of the concentration from its mean in the cross section as a function of distance is calculated. When the steady state is reached, the mixing length may be determined.

Obstacles may be situated in the specified domain for studying the mixing length variations. An obstacle is a region in which the borders fulfill the no-slip conditions (null velocities).

Simulation results

As we said before, the software was developed for studying *non compressible fluids* in two and three dimension domains. The grid used in this case (finite differences discretisation) is a two dimension one, of 150 x 20 cells. A balance was made among

the Reynolds number, cell dimension, time step, stability of the algorithms and computer cost (memory and time demand).

Navier-Stokes equations were discretised, as the convective terms of the momentum equations become dominant at high Reynolds numbers, a weighted mixture of central differences and donor-cell was employed.

A *no-slip condition* was adopted for boundaries and obstacles. The velocities field obtained with the Navier-Stokes equations was used to simulate a convection-diffusion model on it.

The results are shown in Figures 1 and 2, for a given Reynolds number. As a continuous tracer injection is considered, the steady state reached is shown in both figures. The injection is simulated in a portion of 9 cells (x=50 to 52, y=14 to 16).

In Figure 1 the situation without any obstacle is exampled. The good mixing length is achieved in approximately 4 cross sections, as the concentration's standard deviation is below the adopted criteria.

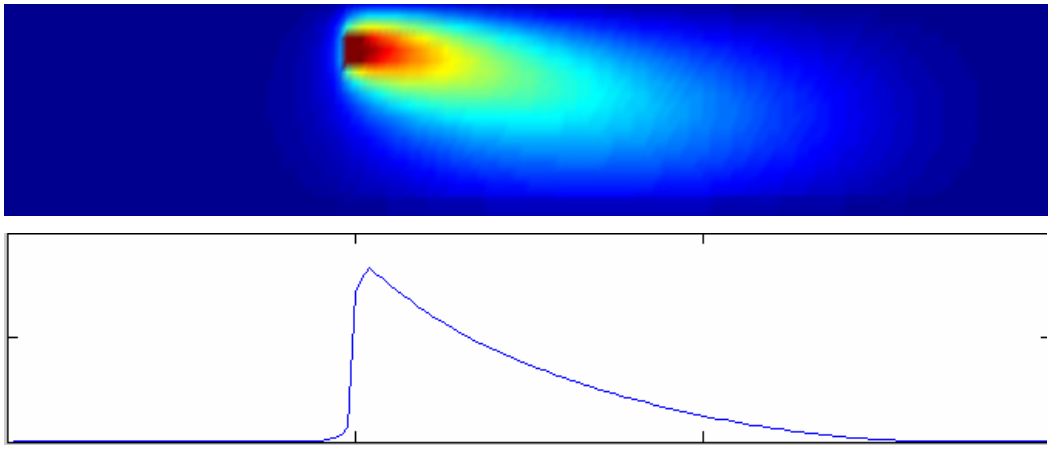

Figure 1.- Tracer distribution and standard deviation (without obstacles).

When an obstacle is considered (the obstacle is asymmetrically situated, at x=65 to 75, y=8 to 10), the good mixing length is shortened to approximately its half. The situation is shown in Figure 2.

No turbulence model is included in the written software, due to its computer cost. Even so, vorticities around the obstacle should be the reason of that mixing length variation.

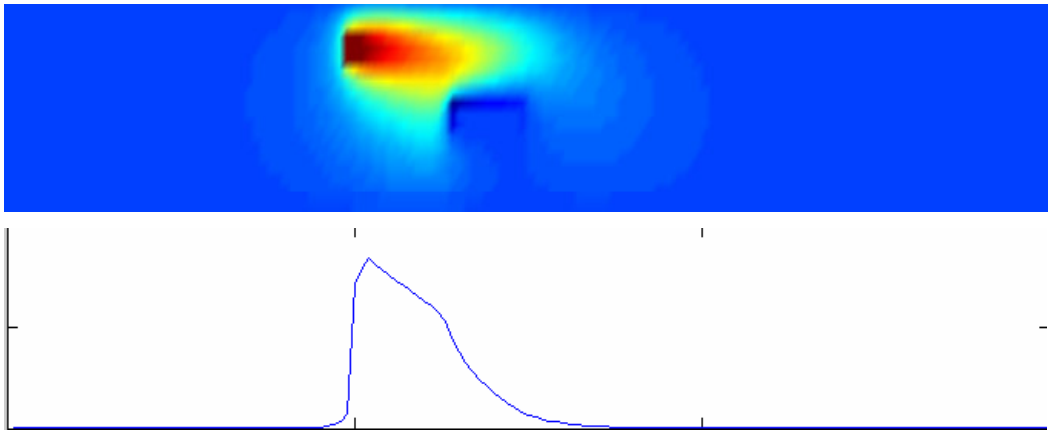

Figure 2 Tracer distribution and standard deviation (with an obstacle).

Discussion and conclusions:

(1)We considered two situations, summarized in Figures 1 and 2. The first one doesn't consider any obstacle. In the second an obstacle is situated near the injection point, within the mixing length. As may be expected, the mixing length is reduced when vortices are created by an obstacle; actually, the good mixing length is shortened to approximately its half. Varying the distance between the obstacle and the injection point, the corresponding values of the mixing length was determined as an increasing function of the aforementioned distance.

(2)A mapping of the *good mixing length* to *Reynolds number should* be studied in both situations, with and without obstacles.

(3) As will be shown elsewhere, the residence time distribution in the point $P \in \Sigma$ far enough ahead an injection point $P_0 \in \Sigma_0$, $h(t, P; 0, P_0)$, for t big enough, differs from the residence time distribution corresponding to good mixing conditions $h_\infty(t, P)$ by a factor that depends of the quotients between an advection-diffusion length scale $L_o = D/v$, and two characteristic length scales $L_c(P)$ of c and $L_I(P)$ of I_c, defined along an averaged and suitably selected flow line.

Bibliography

ANDERSON, J. et al., **Computational Fluid Dynamics**, Mc.Graw Hill, 1995.

GERALD, Curtis F., **Applied Numerical Analysis**, Addison Wesley, 1989.

GRIEBEL, M et al., **Numerical Simulation in Fluid Dynamics: A Practical Introduction**, SIAM, 1997.

VERSTEEG, H.K. et al., **An introduction to Computational Fluid Dynamics, the finite volume method**, Prentice Hall, 1995.