Available online at www.sciencedirect.com
ScienceDirect
ELSEVIER
Applied Mathematical Modelling 33 (2009) 1248–1258
APPLIED MATHEMATICAL MODELLING
www.elsevier.com/locate/apm

Different numerical methods in the study of passive scalar transport in a pipeline *x*-junction

W. Vicente *, M. Salinas-Vazquez, C. Chavez, E. Carrizosa

Instituto de Ingeniería, Universidad Nacional Autónoma de México, Ciudad Universitaria, 04510 México DF, Mexico

Received 1 April 2006; received in revised form 1 January 2008; accepted 23 January 2008
Available online 8 February 2008

Abstract

A computational fluid dynamic model is used to analyze the transport processes of a passive scalar generated in the mixing of two fluids or flows in a pipeline *x*-junction. Turbulent flow field is computed for the merging of streams using two and three-dimensional simulations, which are achieved employing Cartesian coordinates, BFC and a cut cell method. These different numerical solving methods are compared. The numerical model is validated through an experimental set-up. Different parameters are measured for various operating conditions. The influence of the angle between pipe inlets is studied to establish the optimal condition in which the passive scalar concentration in both outlets is similar.

Keywords: Concentration; Pipes; Potable water; *x*-Junction; CFD

1. Introduction

Turbulent mixing of many fluid streams in pipelines is of much interest due to its wide application in industrial and power generation processes. Of particular interest for this work is the study of potable water distribution systems, where water from different sources is mixed. In addition, the dispersion of some disinfectant or polluting substances in the water is also analyzed.

Different *T*-junction type pipelines have been studied numerically and experimentally [1–4]. However, a reduced number of studies are available for a confined cross-flow, which requires a more complex geometry and turbulent mass transfer simulation. The geometry studied here is a two-pipe configuration with an *x*-like crossing point. Both pipes transport clean water, but a specified concentration of a passive scalar is added in one of the inlets.

An experimental set-up was designed for this work, consisting of an *x*-junction with an angle between the two pipe inlets $\alpha = 90°$. The volumetric flow rate in the clean water inlet was varied to study its effect on concentrations in both pipe outlets. The passive scalar used in the experimental set-up represents the hipochlorine

* Corresponding author. Tel.: +52 55 56 23 35x1113; fax: +52 55 56 16 21 64.
E-mail address: wvicenter@ii.unam.mx (W. Vicente).

0307-904X/$ - see front matter © 2008 Elsevier Inc. All rights reserved.
doi:10.1016/j.apm.2008.01.022

in the water. Different parameters were obtained experimentally and results were used for validating the numerical model.

In the numerical model, the hydrodynamics was represented with the Reynolds-averaged continuity, momentum and conservative species equations. The standard $k-\varepsilon$ model [5,6] was used to consider turbulence effects within the flow. The hydrodynamic equations were solved using a finite-volume method.

When flow simulations involve curved geometries or boundaries that are not aligned with the grid orientation, the use of Body-Fitted Coordinates (BFC) or staircase steps approximation in Cartesian coordinates is common. However, many problems are observed, such as: inaccurate results, high computational costs, low accuracy or iteration divergence. As an alternative approach, cut cell methods have become popular, because they have the simplicity and accuracy of the Cartesian coordinates for internal cells (cells with fluid flow), and a more elaborate method only for the boundary surfaces. Many different cut cell methods have been developed, such as: immersed boundary or surface method, embedded boundary method, or the ghost fluid method. In this work, the ASAP method (Arbitrary Source Allocation Procedure) [7,8], which determines the intersections of the complex geometry with the Cartesian grid lines and calculates the free areas and volumes of the partially blocked cells for modifying the transport by source terms, is used.

Considering that numerical simulations give rise to the study of more parameters than those studied experimentally, the present work is only focused on analyzing the influence of the α angle between the pipe inlets in the outlet passive scalar concentrations.

The paper is organized as follows: in the next two sections, the characteristics of the experimental set-up and the numerical model are discussed, and results and performances of different solving methods are compared. Section 4 compares the numerical and experimental results for validation of the mathematical model. In Section 5, the effect of the angle between both pipe inlets over the passive scalar concentration in the outlets is analyzed. Finally, in Section 7, an outline of the most relevant results is presented as conclusion.

2. Experimental set-up

The experimental set-up of the x-junction was designed and constructed in the Hydraulic Laboratory of the Engineering Institute of the National University of Mexico (UNAM). A schematic view of the experimental set-up is shown in Fig. 1. The junction was built in acrylic tube with 90° cross-shaped pipe. The pipe length was 1.23 m with an inner diameter of 0.01905 m (3/4″). The junction had two inlets and two outlets, labeled *inlet1*, *inlet2* and *outlet1, outlet2* respectively. *Inlet1* was supplied with a mixture of clean water and a dye substance, while *inlet2* was delivered with only clean water. Fluid for both inlets was supplied by two storage tanks, provided with one 372.85 W (1/2 HP) pump each. The dye substance was blue ink.

Measurements of volumetric flow rates at the inlets were taken with flow-meters, while at the outlets, volumetric tanks were employed. The concentrations were measured through a Hach spectrophotometer, which employs a color technique in terms of platinum–cobalt units. At the branches, pressure was measured with a piezometer.

Tests were performed for different volume flow rates at *inlet2*. The volume flow rate at *inlet1* was approximately equal to 0.4 dm^3/s for all cases. The Reynolds number (based on mean inlet velocity and pipe diameter) at *inlet1* was $Re = 28{,}000$. The volumetric flow and Reynolds number for *inlet2* varies from 0.30 to 0.80 dm^3/s, and from 23,000 to 50,000, respectively. The experimental uncertainties associated with the measurement of volume flow rates and the concentrations of tracer species were 3% and 4%, respectively.

3. Numerical model

3.1. Numerical procedures

Different procedures were performed to represent the cross-flow. The simplest form is based on a Cartesian grid through the application of porosity, Fig. 2a, where values go from 0, blocked cell, in the solid zone without fluid flow, and 1 in the fluid zone where the entire cell is open to the flow. The second form consists of a BFC grid, where only the flow zone is considered. For this procedure, three different zones were represented and interconnected, Fig. 2b. In Table 1, we present the CPU time and memory used for an example of cross-