CFD for Environment and Civil Engineering applications: Vofor2DV model

Rita F. Carvalho
IMAR, Department of Civil Enginering,University of Coimbra, Portugal
ritalmfc@dec.uc.pt

Abstract

The use of CFD and efficient results visualization is increasing in civil-hydraulics and environment engineering applications, even in areas traditionally based in physical experience like river hydraulics, dams and hydroelectric projects (see http://www.aldenlab.com/index.cfm/news?NID=107).

This presentation aims to present an overview of the fluid equations of motion, simplified equations and auxiliary models most used in civil and environment engineering field and the associated numerical models. A numerical model for studying two-dimensional flows in a vertical plane based on a volume of fluid and a fractional volume and area obstacle representation techniques is also presented emphasizing some details of numerical approaches, auxiliary models, recent developments and potential future refinements.

Starting by the fluid equations of motion, the well known Navier-Stokes equations, details, usual simplifications, as well as the characteristics of the flow behavior in specific cases in these fields are pointed. Therefore, several models like Saint-Venant, Boussinesq type and VOF type are briefly described, emphasizing the main limitations and developments. Some applications that reinforce the aim of each kind of model are presented. Special care is given to the results analysis and their comparisons between physical, model and numerical representations.

The VOFOR2DV model is based on the Reynolds-averaged Navier-Stokes equations governing the motion of the mean 2D incompressible flows in the $x-z$ plane, in which the free surface is described using a refined Volume-Of-Fluid (VOF) method [4]. The internal obstacles are described by means of the Fractional Area-Volume Obstacle Representation (FAVOR) algorithm [5]. The governing equations are discretized using a finite difference staggered grid system of rectangular cells (control volume) with variables width Δx_i and height Δz_j. The porosity function θ is defined at the horizontal and vertical faces of each control volume using the mesh-wide arrays AT and AR and at the centre of the control volume using another array AC allowing the treatment of curved boundaries in cartesian meshes without stair-step instabilities. The details of the VOFOR2DV can be found in [1]. Figure 1 illustrates some hydraulic structures applications.

Numerical simulations using different mathematical models are then compared with some experimental work: a partial submerse break water and landslides into the water. Finally, recent advances related with auxiliary model inclusion in Vofor2dv are presented.

Keywords: CFD, free-surface, numerical model**,** VOF, FAVOR

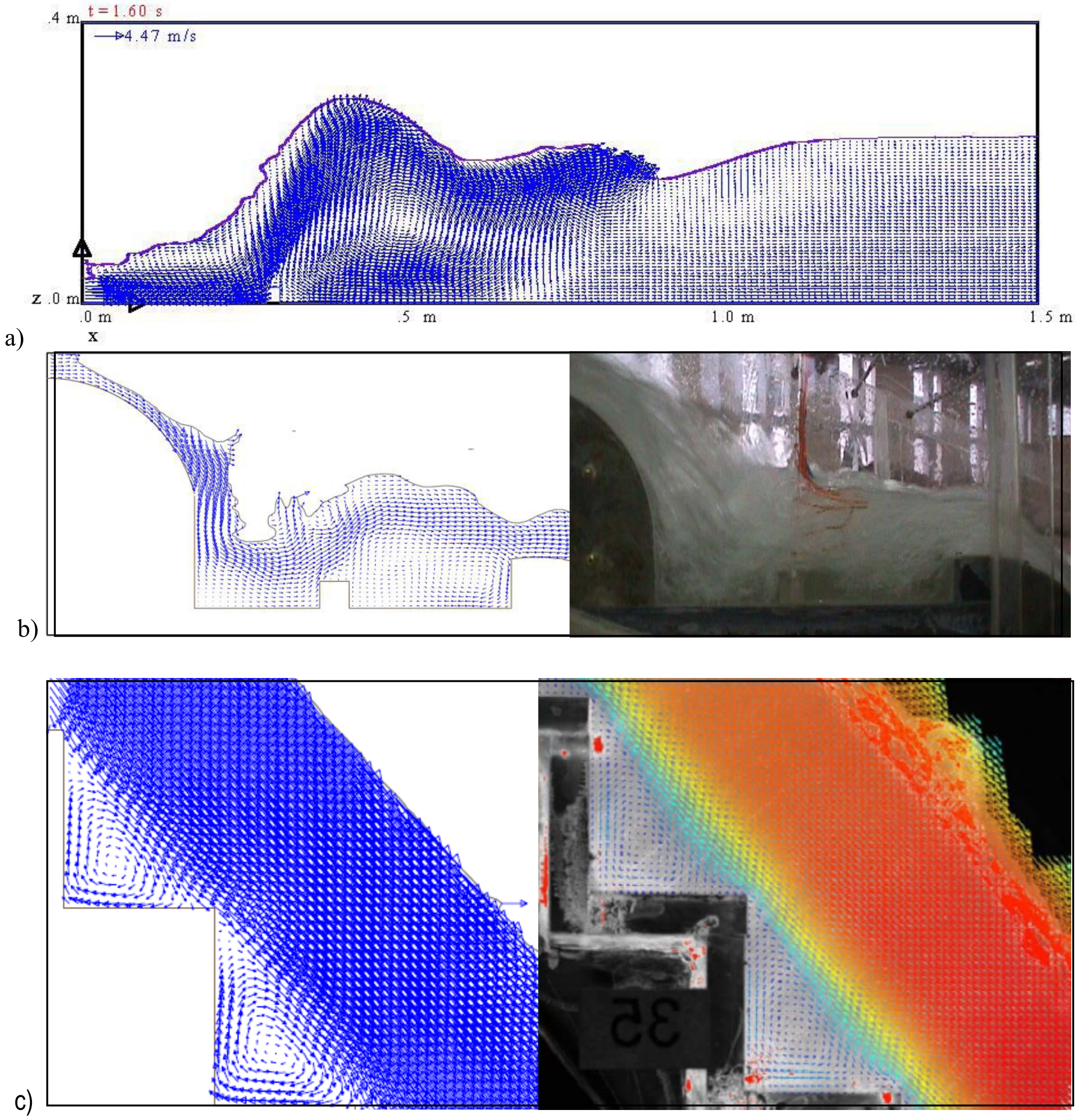

Figure 1: a) Hydraulic jump stilling basin [1]; b) Hydraulic jump stilling basin with a baffle wall [1]; c) Stepped spillway with small slope: numerical results vs photo [3]; d) Stepped spillway with large slope numerical results vs PIV data [2]

References

[1] Carvalho, R. F. 2002, "Acções hidrodinâmicas em estruturas hidráulicas: Modelação computacional do ressalto hidráulico", Tese de Doutoramento, Faculdade de Ciências e Tecnologias da Universidade de Coimbra, Coimbra

[2] Carvalho, R. F. & Amador, A.T., "Flow field over a stepped spillway: Physical and numerical approach", IAHR – Symposium on Hydraulic Structures (IAHR), 12-13 October, 2008, China

[3] Carvalho, R. F. & Martins, R. – "Stepped spillway with hydraulic jumps: scale and numerical models of a conceptual prototype": Journal of Hydraulic Engineering da ASCE (submitted and partial accepted 2007);

[4] Hirt & Nichols, 1981. "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries". J. Comp. Phys. 39, 201–225.

[5] Hirt, C. W. and Sicilian, J., "A porosity technique for the definition of obstacles in rectangular cell meshes", Fourth International Conference on Ship Hydrodynamics, Washington, DC, September 1985.