Mechanics Research Communications 37 (2010) 119–121

Contents lists available at ScienceDirect

Mechanics Research Communications

journal homepage: www.elsevier.com/locate/mechrescom

Numerical simulation of a weak breaking tidal bore

Pierre Lubin [a,*], Stéphane Glockner [a], Hubert Chanson [b]

[a] Université de Bordeaux, TREFLE-ENSCBP UMR 8508 CNRS, 16 Avenue Pey-Berland, 33607 Pessac Cedex, France
[b] The University of Queensland, School of Civil Engineering, Brisbane QLD 4072, Australia

ARTICLE INFO

Article history:
Received 3 September 2009
Available online 4 October 2009

Keywords:
Environmental flows
Navier–Stokes equations
Numerical simulations
Tidal bores

ABSTRACT

A tidal bore is a natural and fragile phenomenon, which is of great importance for the ecology of an estuary. The bore development is closely linked with the tidal range and the river mouth shape, and its existence is sensitive to any small change in boundary conditions. Despite their ecological and cultural value, little is known on the flow field, turbulent mixing and sediment motion beneath tidal bores. Indeed, some striking features can be highlighted in two-dimensional simulations, such as large velocity fluctuations and flow recirculation structures. Using Large Eddy Simulation method, we present numerical results that show the complicated turbulent structures and their unsteadiness under a tidal bore.

© 2009 Elsevier Ltd. All rights reserved.

1. Introduction

A tidal bore is a positive surge propagating upstream as the tidal flow turns to rising in river mouths exhibiting converging funnelled channel forms during low freshwater conditions. The tidal bore is a very vulnerable process and it results from a fragile balance between many parameters (e.g. bathymetry, tidal conditions, etc.). Very few field observations were conducted to date, while some recent laboratory experiments brought new insights to the turbulent motion (Koch and Chanson, 2005, 2009). Nevertheless, the tidal processes remain poorly understood today (Chanson, 2009). A recent numerical model based upon the Navier–Stokes equations (Furuyama and Chanson, 2008) was compared to laboratory experiments (Koch and Chanson, 2005). Some interesting features were observed, but the results lacked a fine mesh grid resolution and accurate numerical schemes.

The goal of our work is to simulate this unsteady two-phase flow tidal bore motion using Large Eddy Simulation method to gain a further understanding of the tidal bore processes. We aim at describing accurately the free-surface behavior and the turbulent flow structure.

2. Numerical model

On a fixed orthogonal curvilinear grid, an incompressible multiphase phase flow between non-miscible fluids can be described by the Navier–Stokes equations in their multiphase form. The governing equations for the Large Eddy Simulation (LES) of an incompressible fluid flow are classically derived by applying a convolution filter to the unsteady Navier–Stokes equations. The velocity/pressure coupling is solved with a pressure correction method (Goda, 1978). The space derivatives of the inertial term are discretized by a hybrid upwind-centered scheme and the viscous term is approximated by a second-order centered scheme (Lubin et al., 2006). The interface tracking is achieved by a Volume Of Fluid method (VOF): a Lax-Wendroff TVD scheme (Total Variation Diminishing) is used to solve directly the free-surface evolutions. A dual grid, or underlying grid (Rudman, 1998), is used to gain an improved accuracy for the interface description, the mesh grid size being divided by two in each direction for the interface tracking. The turbulent viscosity is calculated with the Mixed Scale model (Sagaut, 1998). The numerical model has proved its accuracy for coastal applications (Lubin et al., 2006) and has been benchmarked through numerous test-cases including mesh refinement analysis (Lubin, 2004). The time discretization is implicit and the equations are discretized on a staggered grid thanks to a finite volume method. The MPI library HYPRE is used to solve the linear system of the prediction and correction steps (Falgout et al., 2006).

3. Numerical configuration of a breaking tidal bore

The experimental configuration consists in the generation of a weak positive surge by a rapid partial gate closure at the downstream end of the control volume and its upstream propagation against the initially steady flow (Koch and Chanson, 2005, 2009). The numerical configuration consists in an initial rectangular steady flow motion (from the right side of the numerical domain to the left side) with an initial steady velocity ($V_0 = 1.021\ \mathrm{m\ s^{-1}}$). The initial water depth is $d_0 = 0.0785$ m, as presented in Fig. 1. The two-dimensional numerical domain is 10 m long and 1 m high and is

* Corresponding author. Tel.: +33 (0) 540 003 307; fax: +33 (0) 540 006 668.
E-mail addresses: lubin@enscpb.fr (P. Lubin), glockner@enscpb.fr (S. Glockner), h.chanson@uq.edu.au (H. Chanson).

0093-6413/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.mechrescom.2009.09.008

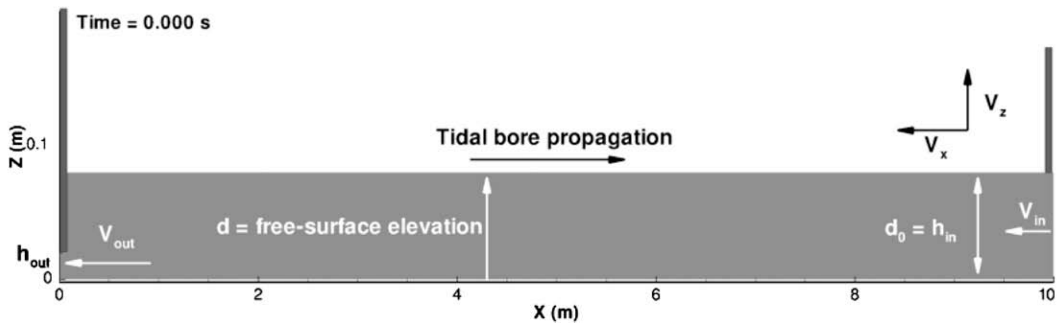

Fig. 1. Sketch of the initial conditions for the tidal bore generation, immediately after the gate closure corresponding to the wall boundary condition set at the left side of the numerical domain.

discretized into 2000 × 1000 regular Cartesian cells. A no-slip condition was imposed at the lower boundary and an open boundary condition is used at the top of the numerical domain. At the left side of the numerical domain, an outlet velocity condition ($V_{out} \simeq 1.76$ m s^{-1}) is fixed to let the water flow below a vertical gate, the outlet height being $h_{out} = 0.02$ m. The inlet velocity ($V_{in} = 1.021$ m s^{-1}) is fixed at the right side of the numerical domain at the inlet height $h_{in} = d_0 = 0.0785$ m. The time step is chosen to ensure a Courant-Friedrichs-Levy number less than 0.1. The calculation is made with the densities and the viscosities of air and water and 128 processors are used.

4. Results

The initial rectangle of water hits the left wall, the water runs-up the wall and splashes down. The generated bore then propagates upstream, towards the right side of the numerical domain. The initial large free-surface deformations are in accordance with the experimental photographs (Koch and Chanson, 2005). The celerity of the bore front is approximatively 0.5 m s^{-1}, as experimentally recorded. We investigated the horizontal and vertical velocity components as functions of time (not shown here). The bore front passage is associated with a rapid flow deceleration, coupled with a sudden increase in water depth. Some flow reversal is observed next to the bed. This was discussed and associated with some transient flow separation and recirculation (Koch and Chanson, 2005, 2009).

Some large structures are shown in Fig. 2: a main recirculation structure observed beneath the bore front and propagating upstream (towards the right side of the numerical domain), and some macroturbulent structures separating for the previous one and propagating downstream (towards the left side of the numerical

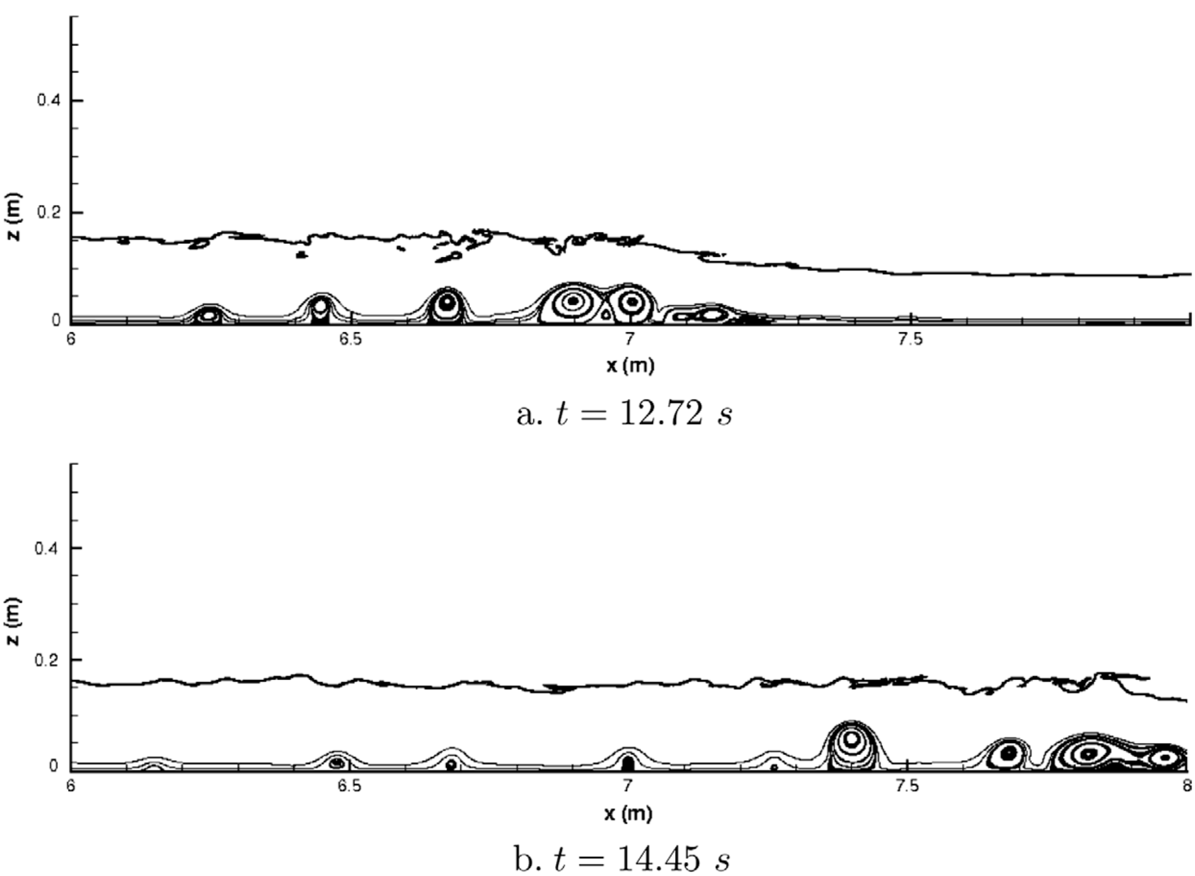

Fig. 2. Streamlines indicating recirculation structures under the propagating tidal bore.

domain). The height of these large eddies is approximately half of the downstream water depth, at the time of their generation. It can be observed that the structures are generated in sequence below the bore front, the continuous process exhibiting some regular temporal and spatial frequencies. The main large flat-shaped recirculation cell appears as the bore front propagates. It then grows in size as the water depth suddenly increases. This cell then splits in structures of similar size. The recirculation structures then separate regularly in time and are released to propagate downstream, spaced every 20 cm from each other. The eddies are advected at about $V_{adv} \simeq 0.3\ \mathrm{m\,s^{-1}}$, which gives a ratio $V_{adv}/V_2 \simeq 0.6$, V_2 being the flow velocity after the bore passage. These vortical structures remain next to the bed as these persisting coherent structures are advected towards the left side of the numerical domain while the breaking bore propagates upstream. This implies that a great amount of sediment could be placed into suspension and transported by the main flow. These original results confirm and illustrate the experimental observations identifying for the first time these recirculations under tidal bores (Koch and Chanson, 2005, 2009).

At this point, these preliminary results are limited by two aspects. First, the numerical configuration is two-dimensional. Then, there is a need for realistic unsteady inflow conditions to be specified at the inlet boundary. Some numerical tests were performed with the basic technique consisting in generating turbulent inflow data by taking the mean experimental velocity profile with superimposed random fluctuations. The generated data do not exhibit any spatial or temporal correlations, and the pseudo turbulence is quickly dissipated. An effective method to generate synthetic eddies on the inlet plane is under implementation (Jarrin et al., 2006).

5. Conclusion and future work

The major result of this ongoing work is the identification of recirculation structures generated in sequence below the front of the propagating tidal bore and advected downstream. The main features of the flow are in accordance with the basic experimental results (weak breaking bore, flow reversal and rapid flow deceleration). Some 3D numerical developments are undertaken to overcome the limitations of the inlet boundary conditions, to confirm these first observations and to investigate more into details the generation process of the recirculation structures.

Acknowledgements

The authors wish to thank the Aquitaine Regional Council for the financial support dedicated to a 256-processor cluster investment, located in the TREFLE laboratory. This work was performed using HPC resources from GENCI-CINES (Grant 2009-c2009026104). Prof. Hubert Chanson also acknowledges the funding for a visiting professorship position at the Université de Bordeaux and a fellowship from the Australian Academy of Science.

References

Chanson, H., 2009. Current knowledge in hydraulic jumps and related phenomena. A survey of experimental results. Eur. J. Mech. B: Fluids 28 (2), 191–210.

Falgout, R.D., Jones, J.E., Yang, U.M., 2006. The design and implementation of hypre, a library of parallel high performance preconditioners. In: Numerical Solution of Partial Differential Equations on Parallel Computers. Springer-Verlag, pp. 267–294 (Chapter 51).

Furuyama, S.-I., Chanson, H., 2008. A Numerical Study of Open Channel Flow Hydrodynamics and Turbulence of the Tidal Bore and Dam-break Flows. Technical Report No. CH66/08, Division of Civil Engineering, The University of Queensland, Brisbane, Australia.

Goda, K., 1978. A multistep technique with implicit difference schemes for calculating two- or three-dimensional cavity flows. J. Comput. Phys. 30, 76–95.

Jarrin, N., Benhamadouche, S., Laurence, D., Prosser, R., 2006. A synthetic-eddy-method for generating inflow conditions for large-eddy simulations. Int. J. Heat Fluid Flow 27, 585–593.

Koch, C., Chanson, H., 2005. An Experimental Study of Tidal Bores and Positive Surges: Hydrodynamics and Turbulence of the Bore Front. Technical Report No. CH56/05, Department of Civil Engineering, The University of Queensland, Brisbane, Australia.

Koch, C., Chanson, H., 2009. Turbulence measurements in positive surges and bores. J. Hydraul. Res. 47 (1), 29–40.

Lubin, P., 2004. Large Eddy Simulation of Plunging Breaking Waves. Ph.D. Thesis, Université Bordeaux I (in English).

Lubin, P., Vincent, S., Abadie, S., Caltagirone, J.-P., 2006. Three-dimensional Large Eddy Simulation of Air Entrainment under Plunging Breaking Waves, vol. 53, pp. 631–655.

Rudman, M., 1998. A volume-tracking method for incompressible multifluid flows with large density variations. Int. J. Numer. Meth. Fluids 28 (2), 357–378.

Sagaut, P., 1998. Large Eddy Simulation for Incompressible Flows – An Introduction. Springer-Verlag.