Some Experiences with Numerical Modelling of Overflows

Larsen, Torben; Nielsen, L.; Jensen, B.; Christensen, E.D.

Published in:
The Fifth International Symposium on Environmental Hydraulics (ISEH V)

Publication date:
2007

Document Version
Publisher's PDF, also known as Version of record

SOME EXPERIENCES WITH NUMERICAL MODELLING OF OVERFLOWS

T. Larsen[1], L. Nielsen[2], B. Jensen[3] and E.D. Christensen[3]

[1] Aalborg University, Civil Engineering, Sohngaardsholmsvej 57, 9000 Aalborg, Denmark, e-mail: torben.larsen@civil.aau.dk
[2]NIRAS A/S, Sortemosevej 2, 3450 Allerød; Denmark, e-mail lni@niras.dk
[3] DHI Water & Environment, Agern Alle 5, 2650 Horsholm, Denmark, e-mail: bjj@dhigroup.com and edc@dhigroup.com

ABSTRACT

Overflows are commonly applied in storm sewer systems to control flow and water surface level. Therefore overflows play a central role in the control of discharges of pollutants from sewer systems to the environment.

The basic hydrodynamic principle of an overflow is the so-called critical flow across the edge of the overflow. To ensure critical flow across the edge, the upstream flow must be subcritical whereas the downstream flow is either supercritical or a free jet. Experimentally overflows are well studied. Based on laboratory experiments and Froude number scaling, numerous accurate and reliable formulas for the estimation of overflows have been derived.

Numerical modelling of overflows is significantly more complicated than standard 1-dimensional river or sewer modelling. The problem is usually managed by incorporating the mentioned empirical formulas in the numerical models. If there are no standard formulas for a specific geometry, physical experiments have to be carried out.

The present study uses laboratory experiments to evaluate the reliability of two types of numerical models of overflows in sewers systems:

1. 1-dimensional model based on the extended Saint-Vernant equation including the term for curvature of the water surface (the so-called Boussinesq approximation)
2. 2- and 3-dimensional so-called Volume of Fluid Models (VOF-models) based on the full Navier-Stokes equations (named NS3 and developed by DHI Water & Environment)

As a general conclusion, the two numerical models show excellent results when compared with measurements. However, considerable errors occur when inappropriate boundary conditions and grid resolutions are chosen. The paper describes the physical and numerical models and summarises the results.

INTRODUCTION AND BACKGROUND

Overflows are commonly applied in storm sewer systems to control flow and water surface level. Therefore overflows play a central role in the control of discharges of pollutants from sewer systems to the environment.

The basic hydrodynamic principle of an overflow is the so-called critical flow across the edge of the overflow. To ensure critical flow across the edge, the upstream flow must be subcritical whereas the downstream flow is either supercritical or a free jet.

The head/discharge relationship for a broad-crested overflow can easily be determined from classic hydraulic theory based on the continuity and the energy equation, simply by assuming that the Froude number is equal to nil. However, in practice overflows are rarely broad-crested. They are most often rather sharp-edged, which creates a significant curvature of the streamlines, which again causes the pressure distribution to be far from hydrostatic. In this case the classic theory fails and more complex considerations are required.

Experimentally overflows are well studied. Based on laboratory experiments and Froude number scaling, numerous accurate and reliable formulas for the estimation of the head/discharge relationship for overflows have been derived.

Standard 1-dimensional river or sewer modelling is based on the assumption of hydrostatic pressure distribution. Overflows are usually incorporated in the numerical models using the mentioned empirical formulas. If standard formulas do not cover the actual geometry for a specific problem, physical experiments have to be carried out.

The objective of this study is to determine to which extent the physical models can be replaced by numerical computations. Only main points and examples are given in this paper; a more complete description is found in Nielsen (2007).

1-DIMENSIONAL MODELLING OF CRITICAL FLOW

A comprehensive study of this subject has recently been carried out by Zerihun (2004) (see also Zerihun and Fenton, 2006).

Standard sewer and river hydrodynamic models are 1-dimensional models based on the continuity equation and the so-called Saint-Vernant equation (SVE) ; the latter can be understood as a special formulation of the general momentum equation. The SVE yields

$$\frac{\partial Q}{\partial t}+\frac{\partial}{\partial x}\left(\beta\frac{Q^2}{A}\right)=-\frac{1}{\rho}\int_A\frac{\partial p}{\partial x}dA-\frac{\bar{\tau}\,P}{\rho}=0 \qquad (1)$$

where Q is the flow, β the velocity distribution coefficient, A the cross-section area, p the pressure, P the wet perimeter, τ the bed shear stress, t the time, and x is the streamwise coordinate.

In standard models where the pressure distribution is assumed hydrostatic, the pressure term can be expressed in terms of the water depth H. For steady state flow in a rectangular channel, (1) reduces to

$$\left(gH-\beta\frac{q^2}{H^2}\right)\frac{dH}{dx}+gH\left(Z'+S_f\right)=0 \qquad (2)$$

where q is the flow per unit width, H the water depth, β the velocity distribution coefficient, Z´ the bottom slope, S_f the friction slope, and x is the streamwise coordinate.

As described in detail by Zerihun (2004), the influence of the non-hydrostatic pressure distribution must be included in the SVE if near-critical flow is to be described correctly. The solution is known as the Boussinesq approximation. The effect of the centrifugal forces and the curvature of the streamlines can be implemented in the SEQ using various simplified assumptions; but leads to varying accuracy when the equations are discretized and solved numerically. As an example, the so-called BTML version (Boussinesq-type momentum equation with linear variation of the centrifugal term) for steady flow in a rectangular channel (Zerihun, 2004) is:

$$\beta\omega_1\frac{q^2}{2}\frac{d^3H}{dx^3}+\beta\omega_1 Z'\frac{q^2}{H}\frac{d^2H}{dx^2}+\left(1+Z'^2\right)\left(gH-\beta\frac{q^2}{H^2}\right)\frac{dH}{dx}+$$
$$\omega_0\beta q^2\left(\frac{Z''}{2}+\frac{Z'Z''}{H}\right)+gH\left(1+Z'^2\right)\left(Z'+S_f\right)=0 \qquad (3)$$

where Z´´ is the bottom curvature and ω_0 , ω_1 are weight factors for the centrifugal forces.

After dicretization equation (3) can be solved numerically by iteration as explained in detail by (Zerihun, 2004). Examples of such computations are compared with experimental results in figures 1 and 2.

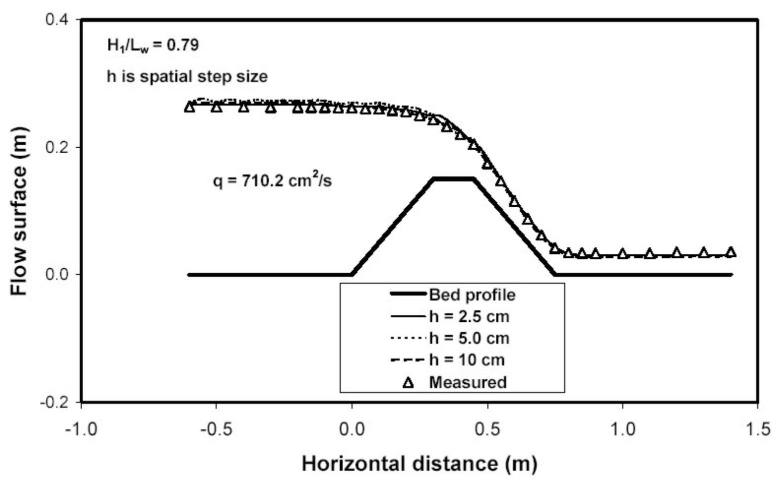

Figure 1 Computed and measured water surface with various spatial resolutions (Zerihun, 2004)

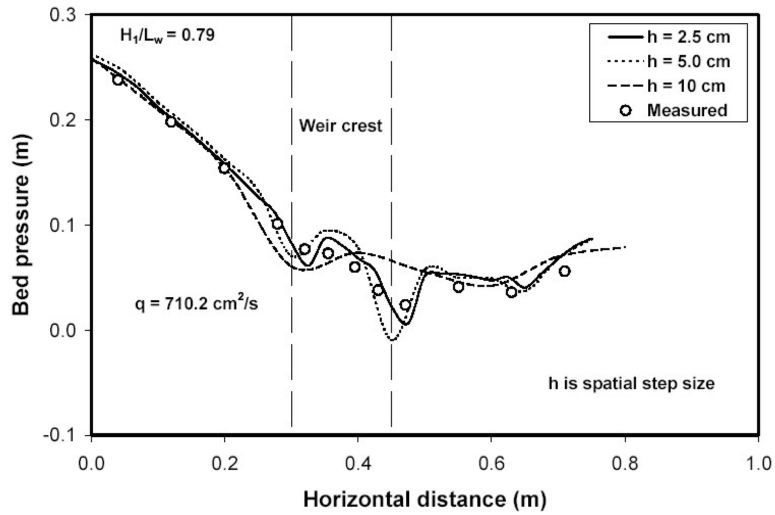

Figure 2 Computed and measured bed pressure with various spatial resolutions (Zerihun, 2004)

The figures indicate clearly that the Boussinesq approximation is well suited for the computation of broad-crested overflows. The results show that the spatial resolution can be rather rough compared to the water depth without introducing larger errors.

The computational procedure in this method is very fast compared to the 2-D and 3-D models mentioned in the following. This suggests the possibility of using the principle in sewer and river models for situations where the flow is close to critical flow; this is relevant for several practical applications.

3-DIMENSIONAL MODELLING OF CRITICAL FLOW

CFD model

The 3-dimensional flow models (often termed CFD-models) are based on the Navier-Stokes equations (NVE), which basically are the three projections of the momentum equations plus the continuity equation. To model turbulent flow we use Reynolds' time-averaging to separate the mean values and the fluctuating parts of velocities and stresses. The NVE is only solved for the mean value of the flow. This time-averaging introduces the so-called Reynolds' stresses in the NVE.

The Reynolds' stresses are then described by a so-called turbulence model, which is here a standard k-ε–model, where the shear stresses are determined from the local turbulent kinetic energy *k* and the dissipation ε of *k*. This is a standard procedure included in most CFD-packages.

Modelling of free surface

CFD-models normally operate with flow within fixed boundaries. The modelling of free surface flow (or 2-phase flow) requires further supplements. A recognized method is the so-called VOF-models (volume of fluid models), which incorporate a moving free boundary. This is done by defining a volume fraction *F* (or degree of filling) in each computational cell, which can take the following values:

$F = 0$	the cell is empty
$F = 1$	the cell is full
$0 < F < 1$	the cell contains the free interface

In order to track the location of the interface, a continuity equation for F is established in the following form

$$\frac{\partial F}{\partial t} + u_i \frac{\partial F}{\partial x_i} = 0$$

The initial condition for the free surface is set by assigning the volume fraction F to be either 1 or 0 in all cells. In principle, the NVE are now solved for one time step in all full cells and a new “field” of F is found, describing the new location of the surface. Various principles for defining the precise location of the surface exist. Details are given in Hirt and Nichols (1981).

NS3 model including VOF

In the present project the NS3 (Navier-Stokes 3-dimensional solver) has been applied. This software was developed by DHI, Institute for Water and Environment, for advanced in-house research in hydrodynamics. The program has several turbulence models available as well as a number of optional VOF formulations. Further details about NS3 can be found in Mayer et al. (1998). The program has been validated against laboratory and full scale measurements in a number of cases, for example reported by Christensen (2006).

The NS3 model always works in transient (non-steady) mode, and the steady state flow situations covered in this study have been computed in this mode by keeping the boundary conditions constant until steady state flow appears.

Comparisons of physical and numerical models

The applicability of the NS3-VOF model was tested by comparing the computed results with the results from two experiments:

1. 2-dimensional experiments with broad crested weirs reported by Zerihun (2004)
2. 3-dimensional experiments of side weirs carried out in own laboratory

The primary objectives of the comparisons were to study the influence of the spatial resolution and to ensure that back-water could not be transmitted upstream through the critical flow zone. A secondary objective was to study the capability of the model when the flow passes through the computational grid under an angle of about 45 degree and with a Froude number just below one on the inlet side of the weir.

2-dimensional models versus experiments

A large number of comparisons between measurements and computations with broad crested, sharp crested, ventilated and non-ventilated overflows were carried out. However, space only allows one example to be presented here, see figures 3 and 4.

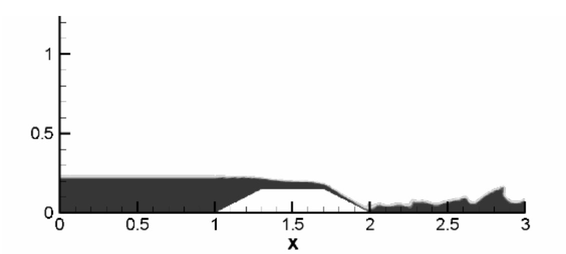

Figure 3 Computation with NS3 of flow over broad-crested overflow

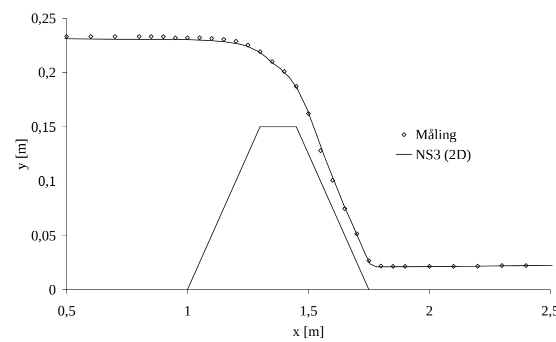

Figure 4 Example of comparison of measurement (måling) and computated (with NS3) water level

The general experience was that the measured and computed water levels deviated less than 5 %, which is close to the accuracy of the measurements. The results also showed that variations in the downstream water level (the downstream boundary condition) did not influence the conditions upstream the point of critical flow.

Running a 2-d model is fast, and the computational time is short. While the practical relevance is limited, the numerical experiments clearly validate the NS3 model with respect to critical flow. This assessment should without doubt also be valid for other CFD packages which includes the VOF assumption for the free surface.

3-dimensional models versus experiments.

In order to assess the numerical model for a 3-D flow situation where the flow direction changes in the area of critical flow, a model of a side weir was set up. The physical and the numerical models are shown in figures 5 and 6.

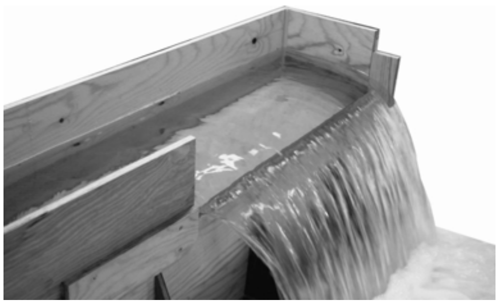

Figure 5 Physical model af side weir

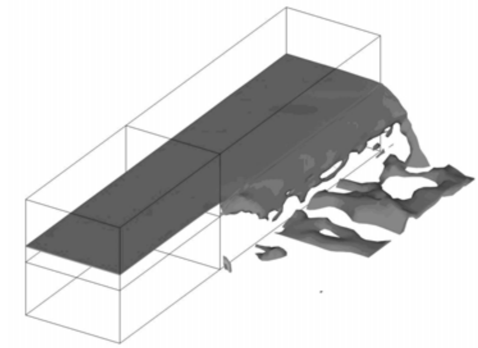

Figure 6 Numerical model of side weir

The comparison between the two models took place in different ways. For example, LDA (laser dobler anemometry) close to the sharp crest was attempted with limited success because of strong fluctuations of the surface here. More successful was the comparison of water surface profiles. An example is given in figure 7.

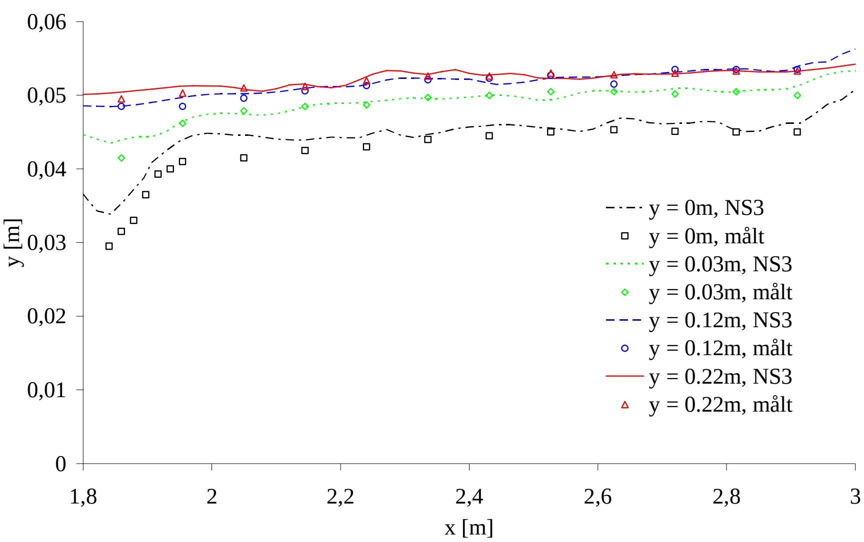

Figure 7 Comparisons of length profiles of water surfaces parallel to overflow crest upstream crest in side weir, y is horizontal distance from crest, NS3 is computed and "målt" is measured.

In general, the comparisons showed excellent agreement. However, in this case (Figure 6) the 3-D modelling was rather time consuming because high spatial and temporal resolutions were needed. The experience was that significantly lower computational Courant number had to be applied compared to the 2-D model. The procedure of finding the optimal resolution proved very time consuming because the execution of each run took in the order of days on a fast PC.

The various turbulence models were tested. Included were computations without any internal friction (no turbulence model) and, as expected, no significant variation was seen.

CONCLUSION

1-, 2- and 3-dimensional hydrodynamic models have been evaluated with respect to their capability for simulating overflows. The 2- and 3-D models are based on direct geometric similarity, whereas the 1-D models are rather simplified representations of reality.

The results have shown that the 1-D models developed by Zerihun (2004) in many cases lead to accurate and reliable results. It seems obvious that these 1-D models could have a future in connection with the 1-D commercial sewer and river models on the market because of their capability of computing critical flows.

For the 2-D and 3-D simulations the NS3 program developed by DHI Water & Environment was applied. NS3 is based on the solution of the full Navier-Stokes equations plus a VOF method for the free surface description. Also here accurate and reliable results were obtained if the necessary spatial and temporal resolutions were chosen. It was found that the type of turbulence model is unimportant for this type of flow.

It can be concluded that numerical modelling in many cases can replace physical model tests when it comes to the design of overflows with complex geometries. On the other hand, it should be kept in mind that the setting up, testing and execution of a numerical model, including finding the optimal spatial and temporal resolutions, can take just as long time as making the physical experiments.

REFERENCES

Christensen, E.D. (2006). Large Eddy simulation of Spilling and Plunging Breakers. Coastal Engineering 53, 463-485

Hirt, C.W., Nicholls, B.D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. J. Comput. Phys., 39, 201-221

Mayer, S., Garapon, A., Sørensen, L. (1998). A Fractional Step Method for Unsteady Free-Surface Flow with Application to Non-linear Wave Dynamics. Int. J. Numer. Meth. Fluids 28, 293-315.

Nielsen, L. (2007). Physical and numerical modelling of overflows (in Danish). MsC thesis, Dept. Civil Eng., Aalborg University.

Zerihun, Y.T. (2004). A One-dimensional Boussinesq-type Momentum Model for Steady Rapidly Varied Open Channel Flows. PhD thesis, The University of Melbourne.

Zerihun, Y.T., Fenton, J.D. (2006). One-dimensional simulation model for steady transcritical free surface flow at short length transitions. Advances in Water Resources 29, 1598-1607.