# CHAPTER 11

## Numerical Modelling of Waves and Currents with regard to Coastal Structures

M. Fischer
J. Juhl
E.B. Rasmussen[1]

### Abstract

This paper describes a two-dimensional numercial model capable of simulating non-stationary flows. Special emphasis has been put on wave motion on and in porous structures, e.g. a rubble mound breakwater. Comparisons of numerical simulations with analytical solutions and model test results have confirmed the applicability of this model for studies of waves and currents with regard to coastal structures.

### Introduction

In the past coastal structures such as breakwaters mainly have been studied by means of physical modelling and simplified numerical calculations. Recent developments in numerical techniques and methods, however, have implied that advanced numerical tools may be adopted in such studies. These numerical models dedicated to coastal structures are still in their infancy but likewise other branches of the hydraulics it is envisaged that numerical models will play an increasing role in future studies.

In the present paper, a special 2D (x-z) version of Danish Hydraulic Institute's three-dimensional model is described. Details on the three dimensional model

---

[1] Danish Hydraulic Institute, Agern Allé 5, DK-2970 Hørsholm, Denmark

adapted here are given in Rasmussen et al. The x-z version is designed especially for flow with regard to coastal structures and porous media. The numerical model is based on the Reynolds-averaged Navier-Stokes equations and the equation for conservation of mass. The equations are discretized into a finite difference scheme imposed on a rectangular, space-staggered grid. The finite difference equations are solved through a non-iterative ADI (Alternating Directions Implicit) technique using the artificial compressibility method.

The energy loss due to both laminar and turbulent effects in porous media is included through the Forchheimer equation. Furthermore, an inertia term has been included in the Forchheimer equation for the case of non-stationary flow.

The free surface boundary in the model has been described applying a subgrid modelling in which the instantaneous position is calculated for each time step by use of linearized momentum equations and kinematic boundary conditions. This implies that the computational domain varies from time step to time step.

The numerical model is applicable to a large range of both dynamic and stationary flow problems with regards to coastal structures such as flows in breakwaters consisting of layers with different porosity, flows through and/or beneath dams and stability of slopes etc. protected by impermeable surface layers likewise.

Model simulations have been compared with both analytical solutions and physical model tests.

## Description of Flows in Porous Media

It is common to apply a macroscopic point-of-view of a porosity layer by describing the porous matrix through characteristic constants. These properties are related both to the fluid and to the granular material in order to describe the penetration of the fluid. This implies that the basic problem is reduced to establish a relation between the pressure gradient and the bulk velocity.

It can be argued as to whether this description is suitable or a microscopic point-of-view is needed. However, such an approach would imply the necessity of a description of each stone with connected geometry and roughness factors. Furthermore, highly sophisticated turbulence descriptions would be required. This leads to unrealistic demands to both model set-up for a simulation of flow in porous media and to the performance of the model itself, since the computational grid should be very fine in order to produce the required resolution of the geometry.

A breakwater normally consists of three porous layers, i.e. core, filter, and armour layer. This implies the necessity of a porosity description, in which multiple layers with different properties can be specified. Physical model tests have shown the necessity of a description of the energy dissipation including both laminar and turbulent flow as well as energy dissipation due to dynamic effects.

The relation between the bulk velocity, u, and the pore velocity, V, is given by

$$u = V \cdot n$$

where n is the porosity.

## Forchheimer equation

The Forchheimer equation consists of two terms expressing the hydraulic gradient due to both laminar and turbulent flow, respectively

$$i = a \cdot u + b \cdot u^2$$

where,

- i is the hydraulic gradient
- a is the laminar dissipation factor
- b is the turbulent dissipation factor

Since the linear term, a, accounts for the laminar effects, it depends on the viscosity. The non-linear term, b, represents the fully turbulent flow and is only dependent on the granular matrix material.

Several relationships of a and b have been proposed in the literature, of which many have been based on a dimensional analysis. In the presented model the relationship proposed by Engelund has been adopted. The laminar and turbulent dissipation terms are described by the constants a and b:

$$a = \alpha \cdot \frac{(1-n)^3 \cdot \nu}{n^2 g d^2}$$

$$b = \beta \cdot \frac{(1-n)}{n^3 g d}$$

where
$\nu$ is the viscosity of the fluid
g is the gravity
d is the stone diameter
$\alpha$ is an empiric constant
$\beta$ is an empiric constant

The formulation of the hydraulic gradient presented above is only valid for a steady state flow. A model for unsteady flow would be to add a time dependent term to the Forchheimer expression

$$i = a \cdot u + b \cdot u^2 + c \cdot \frac{\partial u}{\partial t}$$

The factor c can be expressed in the following way:

$$c = \frac{\left(1 + \gamma \cdot \frac{(1-n)}{n}\right)}{g}$$

where $\gamma$ is the inertia coefficient.

Implementation of porosity description

The model presented solves the Reynolds-averaged Navier-Stokes equations and the continuity equation in a staggered finite difference grid. The prognostic variables are the three velocity components together with the fluid pressure. The adopted porosity description is based on macro parameters of porosity, stone size and dissipation factors. The implementation of this macro scale porosity description involves two changes to the original balance equations

1) Redefinition of terms including velocity with respect to the influence of the porosity.

2) Adding of dissipation terms due to the microscopic flow resistance, i. e. flow between stones. The expression given by Forchheimer together with an additional term for the dynamic effect is applied.

The continuity equation reads

$$\frac{1}{\rho \cdot c_s^2} \cdot \frac{\partial p}{\partial t} + \nabla \cdot V_i = 0$$

where,

$\rho$ is the density
$c_s$ is the speed of sound
p is the excess pressure
V is the pore velocity

The momentum equation reads after introduction of the bulk velocity:

$$c \cdot \frac{\partial u_i}{\partial t} + \frac{1}{n^2} \cdot u_j \cdot \frac{\partial u_i}{\partial x_j} =$$

$$-\frac{1}{\rho} \cdot \frac{\partial p}{\partial x_i} - g_i - g \cdot a \cdot u_i - g \cdot b \cdot |u_i| u_i + \frac{1}{n} \cdot \frac{\partial}{\partial x_j}\left(E \cdot \frac{\partial u_i}{\partial x_j}\right)$$

where E is the eddy viscosity

## Free Surface Description

The applied free surface description of waves is presented in the following. The method is inspired by the VOF method proposed by Nichols and Hirt but splits the volume fraction into space increment fractions in the three coordinate directions, and can as such be considered as a surface tracking method rather than a volume tracking method.

The presented description includes three dependent variables in addition to the velocity components and the fluid pressure. The variables noted $\alpha$, $\beta$ and $\gamma$ represent fractions of space increments in the x-, y- and z-direction, respectively, and thus describe the location of the free surface within the current grid cell, see Fig. 1. In the present model the instantaneous position of the water is directly calculated, which is the main difference to the VOF method. The fraction of volume in each cell can be found as

$$V = \alpha \cdot \beta \cdot \gamma$$

A free surface cell is identified as a cell containing a non-zero value of V and having at least one neighbouring cell that contains a zero value of V. Cells with zero V values are empty cells whereas cells with non-zero V values are treated as full or interior fluid cells.

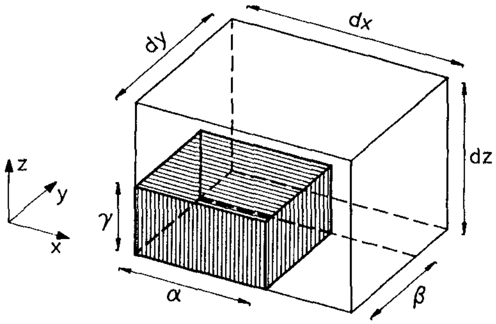


Fig. 1 Description of the free surface by a fraction of volume of fluid technique. The corners represent pressure nodes. For the two-dimensional description $\beta$ = dy.

Briefly, the basic procedure for advancing a solution in time consists of three steps:

1) From the previous time step the dependent variables form the basis for a new discretisation of the conservation of mass and the conservation of momentum equations. The system is solved implicitly taking into account closed boundaries, open boundaries and free surface boundaries.

2) By use of the fractions calculated in the previous time step and on the basis of the newly found dependent variables the fractions $\alpha$, $\beta$ and $\gamma$ are computed.

3) Finally, the fractions defining fluid regions must be used to update the fluid location taking into account the fluid in the adjacent cells and the boundaries of the computational domain.

The theory presented in the following is developed in three dimensions. For reasons of simplicity the implementation of the free surface into the three-dimensional model has been done in two dimensions only - one horizontal and one vertical direction.

The Continuity Equation

In general the continuity equation reads

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}(u\rho)+\frac{\partial}{\partial y}(v\rho)+\frac{\partial}{\partial z}(w\rho) = 0$$

where $\rho$ is the density and u, v and w are the velocity components.

In order to obtain a hyperbolicly dominated system the pressure is introduced into the continuity equation through an equation of state.

$$\frac{1}{\rho c_s^2}\cdot\frac{\partial p}{\partial t}+\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z} = 0$$

where $c_s$ is the speed of sound and p the excess pressure. In the top layer of the computational domain a cell may not be full of fluid. To obtain the continuity equation for the computational cell at the surface an integration over the fraction of fluid volume is done:

$$\frac{1}{\alpha\,\beta\,\gamma}\int_0^{\gamma}\int_0^{\beta}\int_0^{\alpha}\left(\frac{1}{\rho c_s^2}\cdot\frac{\partial p}{\partial t}+\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)dx\,dy\,dz = 0$$

The result of this integration is the continuity equation described in terms of the fractions of volume

$$\frac{1}{\rho c_s^2}\cdot\frac{\partial p}{\partial t}+\frac{\partial u}{\partial \alpha}+\frac{\partial v}{\partial \beta}+\frac{\partial w}{\partial \gamma} = 0$$

The compressibility of the fluid is expressed by the speed of sound $c_s$. In order to make the coefficient matrix of the system diagonally dominanted, an artificial value of $c_s$ should be used.

The Momentum Equations

The conservation of momentum reads:

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i \cdot u_j)}{\partial x_j} = -\frac{1}{\rho} \cdot \frac{\partial P}{\partial x_i} + g_i + \frac{\partial}{\partial x_j}\left(E \cdot \frac{\partial u_i}{\partial x_j}\right)$$

where u is the velocity, P the total pressure, $\rho$ the density, g the gravity and E the eddy viscosity of the fluid.

For reasons of simplicity regarding the space and time discretisation only the linear momentum equations are modelled in the surface cells.

The applied momentum equation for a cell containing a free surface in the x-direction reads

$$\frac{\partial u}{\partial t} = -\frac{1}{\rho} \cdot \frac{\partial}{\partial x}(p+\rho g h)$$

where h is the local, vertical distance to the surface.

Wave Boundary Condition

In order to make simulations of wave impacts on coastal structures an open boundary condition forming propagating waves in the simulation area has been developed. The wave boundary is a mixture of the general Dirichlet type boundary conditions of velocity and level boundaries in the sense that both the level and the velocity are specified. This is presently done by applying a first order wave theory.

## Verifications and Simulations

A number of simulations have been performed in order to verify and study the applicability of the model. A few examples are shown in the following:

### Simulation of Steady State Flow in Porous Media

Verification of the porosity description in the case of steady state flow is carried out by a comparison to experimentals made by Burcharth. The characteristic flow properties, such as the hydraulic gradient and the discharge velocity, have been measured in the case of penetration of water through three different gravel materials. For all three cases the principle model set-up both for the experimentals and for the numerical simulations is shown in Fig. 2.

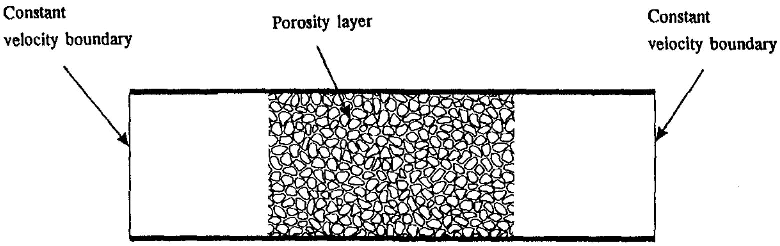


Fig. 2 Model set-up for steady state flow through a porous layer.

The comparison is done in accordance with the following description:

1) Since the hydraulic gradient is given as

$$i = a \cdot u + b \cdot |u| \cdot u$$

where,

| | |
|---|---|
| i | is the hydraulic gradient |
| a | is the laminar dissipation factor |
| b | is the turbulent dissipation factor |
| u | is the bulk velocity |

a straight line is expected when (i/u) is plotted against u. The slope of the line equals b and the intersection with the (i/u)-axis equals a.

2) For the experimentals a and b are deduced as described above. In accordance with the Forchheimer expression and by use of the properties of the gravel material measured by Burcharth the dissipation factors $\alpha$ and $\beta$ are deduced. For the case of steady state flow the dynamic dissipation term equals zero.

3) With the properties of the gravel material and the fluid, simulations of the flow through a porous layer is carried out. Velocity boundaries with a constant value are imposed at both ends of the model area. For each of the three gravel materials the boundary velocity is varied in order to obtain a suitable number of points. The simulations are made with "full slip" closed boundaries, which implies that the pressure gradient is zero outside the porous layer. For all the simulations the kinematic viscosity of the fluid equals $1.34 \cdot 10^{-6}$

$m^2/s$, which is in accordance with the viscosity of the water used by Burcharth.

4) The comparisons of the experimentals and the numerical simulations for two of the gravel materials are shown in Fig. 3.

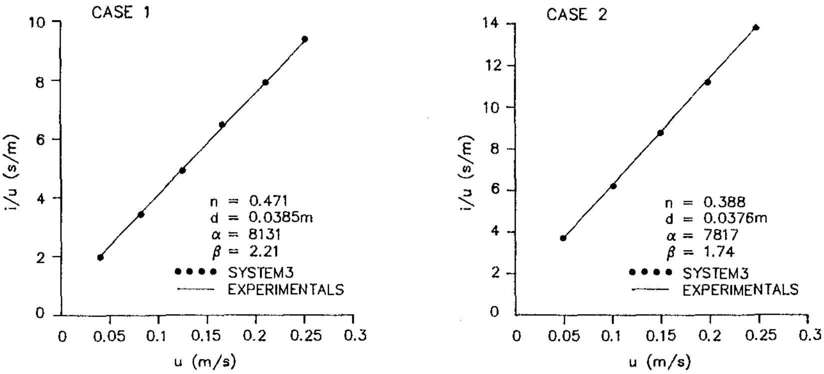


Fig. 3 Comparisons of experimentals and numerical simulations for two cases of steady state flow.

The comparisons show that the model, including a bulk description of the porosity layer, is able to reproduce the measurements for the case of steady state flow.

Simulation of a dam break

Testing of the free surface description is done by simulation of a dam break. Initially a column of water is confined between two vertical walls. When the calculation starts the right wall is removed, and gravity forces the fluid to propagate along the dry floor.

At the beginning of the simulation the fluid is described by 20x20 cells with a size of 0.1 m in both the vertical and horizontal direction. The applied time step is 0.01 sec. Examples of results showing the fluid position and the velocity is presented in Fig. 4.

Experimental results for a dam break test case have been reported by Martin and Moyce and form a basis for a comparison to the model generated results.

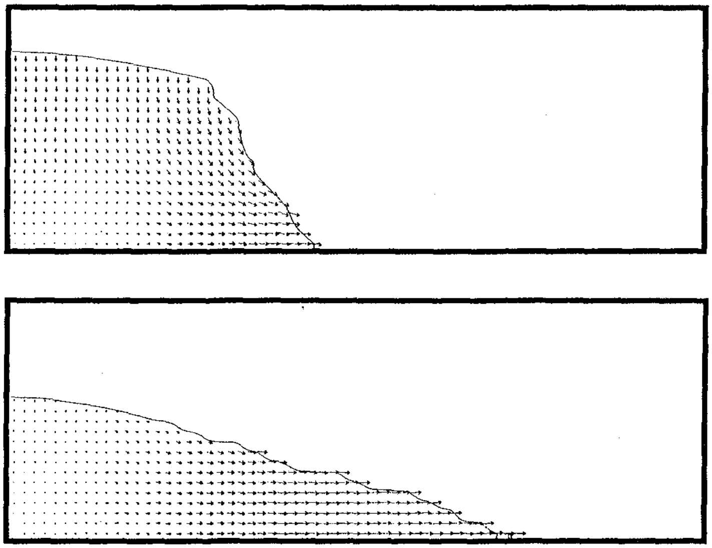

Fig. 4. Example of results for the dam break test. The plots represent the surface location and the velocity for each grid node at time 0.3 sec. and 0.7 sec.

A comparison between model generated results and the experimental results of the toe position vs time is shown in Fig. 5. The largest deviation from the experimental results is everywhere less than one grid spacing.

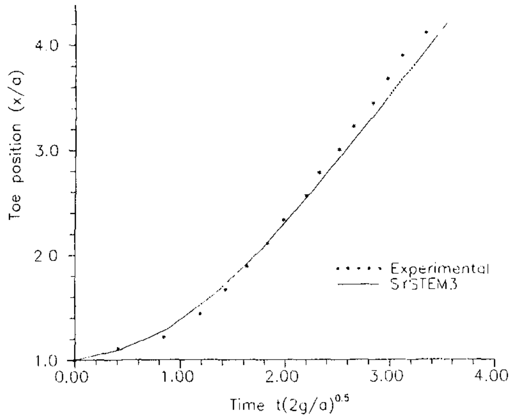


Fig. 5 Comparison of a numerical simulation with experimental data for a dam break.

Wave Run-Up on a Permeable Structure

The combination of porosity layers and a free surface is tested by simulation of wave run-up on a rubble mound breakwater. The breakwater has a sea side slope of 1:2.0 and consists of three porosity layers with the following characteristics:

a = 14.4, 4.9, 2.0 s/m
b = 1820.0, 109.0, 50.0 ($s^2/m^2$)
c = 0.0, 0.0, 0.0
n = 0.35, 0.37, 0.39

The model grid consists of 100 x 3 x 50 nodes and the general parameters of the simulation are:

$\Delta x = \Delta y = \Delta z = 0.015$ m
$\Delta t = 0.002$ sec

At the right end of the model area a wave boundary is applied with the following parameters:

H = 0.06m, T = 1.0s

The still water depth for the simulation is 0.3 m.

An example from a model simulation is shown in Fig. 6. Time series plots of horizontal velocities in three points (as defined in Fig. 6) are shown in Fig. 7.

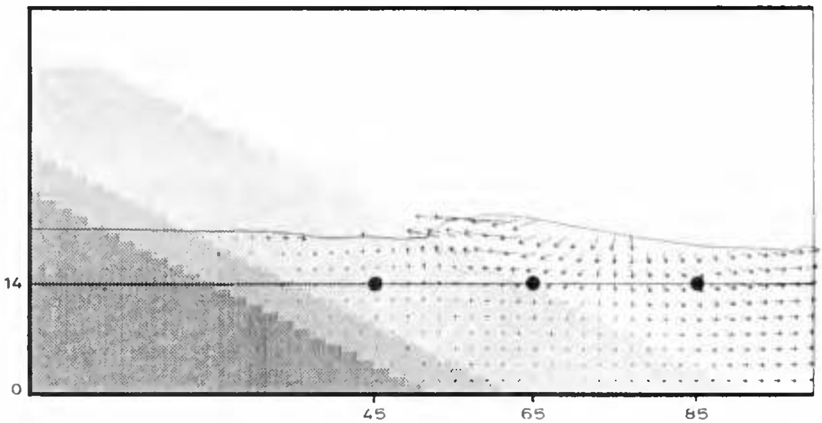


Fig. 6 Example of surface position and velocities during wave run-up on a permeable breakwater. After t = 1.8 s.

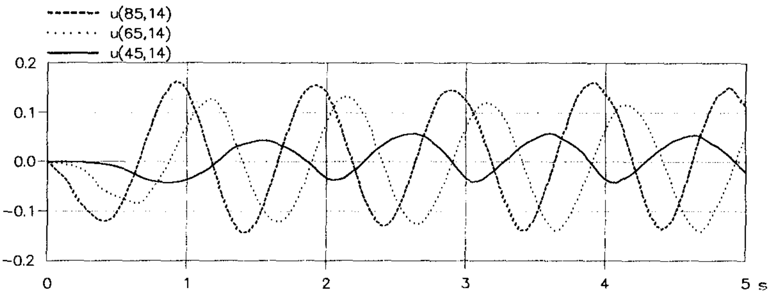


Fig. 7 Time series of horizontal velocities (m/s) at three locations, one outside the breakwater, one at the edge of the breakwater and one in the coarsest porosity layer as shown in Fig. 6.

## Conclusions

A 2D (x-z) numerical model has been developed for description of flows on and in coastal structures. The model includes a description of the energy loss in porous media taking into account both laminar and turbulent effects as well as the inertia effect. Comparisons with analytical solutions and measurements from physical model tests with waves and currents have shown promissing results.

In order to correctly simulate the flow on and in porous coastal structures, it will be necessary to establish a better knowledge of the coefficients involved in the energy loss equation and to describe the energy loss due to wave breaking on a slope which implies a formulation of the hereby induced air entrainment.

## Acknowledgement

The research project which started in 1988 has been funded by the Danish Council of Technology through grant no. 880440. The verification of the porosity description has partly been co-sponsored by the Commission of the European Communities under the MAST-programme, contract 0032.

## References

Burcharth, H.F. & Christensen, C.
"On stationary and non-stationary porous flow in coarse granular materials", University of Aalborg, Denmark (1991).

Engelund, F.
"On the laminar and turbulent flows of ground water through homogenous sand", Trans. Danish Academy Tech. Science, Vol. 3 (1953).

Martin, J.C. & Moyce, W.J.
"An experimental study of the collapse of liquid columns of a rigid horizontal plane", Phil. Trans. Roy. Soc. London A244, 312 (1952).

Nichols, B.D., Hirt, C.W., Hotchkiss, R.S.
"SOLA-VOF: A solution algorithm for transient fluid flow with multiple free boundaries", Los Alamos Scientific Laboratory Report LA-8355 (1980).

Rasmussen, E.B., Vested, H.J., Justesen, P. & Ekebjærg, L.C. "System 3 - A Three-dimensional Hydrodynamic Model", Danish Hydraulic Institute, Denmark (1990).