Hydrodynamics around coastal structures

J.S. Antunes do Carmo & J.L. Carreiras

1 Department of Civil Engineering, University of Coimbra, Portugal
2 Polytechnic Institute of Tomar, Portugal

Abstract

The numerical model MECCA (Model for Estuarine and Coastal Circulation Assessment) – which was developed to simulate tidal currents, density currents or those generated by wind action, in bays or coastal zones (Hess, 1989) – is a “three-dimensional model”, or a quasi-3D model, which is able to satisfactorily simulate complex three-dimensional flows. The MECCA model algorithm is based on the continuity equation (under the non-compressibility hypothesis), on the Navier-Stokes equations (considering hydrostatic equilibrium and the Boussinesq hypothesis) and on conservation equations for temperature and salinity. It is structured in two modes: the external-mode, obtained by vertical integration of the Navier-Stokes equations, and the internal-mode, obtained by subtracting the equations for the external-mode velocity from the equations for the total velocity. The flow within the bottom boundary layer is alternatively described by a Prandtl type model or by a IDV model of the K-L type.

We conducted a rigorous study of its characteristics and hydrodynamic capacities with the aim of extending the MECCA model to sedimentary dynamics applications (suspended and bedload transport). Numerical results were validated by comparisons with laboratory data and other numerical results and theoretical solutions.

This work presents various simulations of the flow around structures commonly occurring in the fluvial and coastal environment, like structure pillars, breakwaters and piers. Particular attention has been given to the two-dimensional and three-dimensional structures of the flow characteristics. We also present sensitivity analyses of the model, using different grids, as well as the two implemented boundary layer models.
1 Introduction

The erosion and sediment transport estimation around structures commonly found in the fluvial and coastal environment, like bridge piers, groynes and breakwaters, are a major concern for designing these structures and for considering preventive measures. Several empirical formulations, generally resulting from studies conducted on physical models, are frequently utilized to estimate the maximum erosion depth in the region around the obstacles. These formulations, however, do not accurately describe the mean flow, and so when they are applied to complex flow patterns they frequently show bad or inaccurate results. Although they currently entail high computational costs, numerical methods are being proved to be a good alternative, or complement, to the study of hydrodynamic phenomena and sedimentary dynamics. The last few years have seen both a deepening of our knowledge of these phenomena and the development of various numerical models that can simulate complex three-dimensional flows satisfactorily. In general, these models calculate the three velocity field components, but they assume that the pressure is hydrostatic and frequently consider the bottom stresses calculated by empirical laws, which are only valid for steady flows. These computational models are known as 2.5DH or quasi-3D models. In practice, this reduced theoretical rigour is normally irrelevant in comparison with all the other uncertainties found in any similar numerical model application.

Groynes and breakwaters are structures frequently utilized in the Coastal Engineering field, namely, in beach feeding and coastal margin protection, and also to resolve navigation problems. Indeed, understanding the flow characteristics around obstacles, particularly behind them, where lee wake vortices can occur, is essential for designing coastal structures.

The three-dimensional flow characteristics around the base of vertical cylinder piles have two basic structures: the horseshoe vortex, developed in the boundary layer on the bed, at the front; and the lee wake vortices caused by the boundary layer separating over the cylinder surface. These wake vortex patterns are the structures mainly responsible for transporting sediment around the obstacles.

The main aim of this work was to study the application of a quasi-3D numerical model to simulate the flow around a groyne and a pile, and to analyse its behaviour, bearing in mind its extension to sedimentary dynamics.

We have chosen the MECCA (Model for Estuarine and Coastal Circulation Assessment) model, which was developed by the National Environmental Satellite, Data, and Information Service, USA. This model was designed to simulate tidal currents and density-driven currents, or those generated by wind action, in bays and on the shallow continental shelf (Hess, 1989).

2 Numerical model

The MECCA algorithm is based on the continuity equation (under the non-compressibility hypothesis), on the Navier-Stokes equations (considering hydrostatic equilibrium and the Boussinesq hypothesis) and on conservation equations for temperature and salinity. It is structured in two modes: the external...
mode, obtained by vertical integration of the Navier-Stokes equations, and the 
internal mode, obtained by subtracting the equations for the external-mode 
velocity from the equations for the total velocity. The flow within the bottom 
boundary layer is described either by a Prandtl type model or by a 1DV model 
of the K-L type.

2.1 Hydrodynamic model equations

The MECCA model equations consist of two horizontal velocity equations, the 
hydrostatic approximation, an equation of continuity, an equation of state, and 
conservation equations for salinity and temperature. The basic hydrodynamic 
equations used to describe the three-dimensional flow are:

\[ \frac{\partial u}{\partial t} + \frac{\partial uu}{\partial x} + \frac{\partial uv}{\partial y} + \frac{\partial uw}{\partial z} = - \frac{1}{\rho} \frac{\partial P}{\partial x} + f v + \]
\[ + 2 \frac{\partial}{\partial x} \left( u_s \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( u_s \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right) + \frac{\partial}{\partial z} \left( u_s \frac{\partial u}{\partial z} \right) \]

\[ \frac{\partial v}{\partial t} + \frac{\partial vu}{\partial x} + \frac{\partial vv}{\partial y} + \frac{\partial vw}{\partial z} = - \frac{1}{\rho} \frac{\partial P}{\partial y} - f u + \]
\[ + \frac{\partial}{\partial x} \left( v_s \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right) + 2 \frac{\partial}{\partial y} \left( v_s \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left( v_s \frac{\partial v}{\partial z} \right) \]

\[ \frac{\partial P}{\partial z} + \rho g = 0 \]
\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \]

where \( u, v, w \) are the components of fluid velocity in the \( x, y, \) and \( z \) directions, respectively, \( P \) is the fluid pressure, \( u_s \) and \( v_s \) represent the horizontal and 
vertical components of viscosity, \( \rho \) is the water density, \( f \) is the Coriolis 
acceleration, and \( g \) is the gravitational acceleration. A simplified equation of 
state for sea water is \( \rho = \rho_s \left[ 1 + F_p (S, T) \right] \), where \( \rho_s \) is a reference density, \( S \) is 
the salinity (parts per thousand), \( T \) is the temperature (°C), and 
\( F_p = 7 \times 10^{-5} + 8 \times 10^{-4} S - 3.5 \times 10^{-4} T - 2 \times 10^{-6} ST - 4.7 \times 10^{-5} T^2 \).

Equations for the dissolved salt and heat conservation are also included, but as 
we are only interested in hydrodynamic problems, these are not shown.

Equations (1) to (4) are then transformed into a new (sigma) coordinate system 
that has the same upper and lower limits at all locations. The vertical coordinate 
\( \sigma \) is defined as \( \sigma = (-h + z)/H \), where \( h \) is the water surface elevation above mean 
sea level and \( H = H(x, y) \) is the total local depth. The transformation is 
accomplished by substituting the equations in the \( x, y, \) \( \sigma \)-system for the 
corresponding expressions in the \( x, y, z \)-system. The total horizontal velocity is
next decomposed into the vertical average and the departure from the vertical average. This separation allows the vertical average (the external mode) to be updated more frequently than the departure (the internal mode), because the external mode horizontal velocity is closely tied to variations in the water level and therefore changes more rapidly than the internal mode, during the passage of long gravity waves. The external mode velocity is equal to the integrated value of the flow over the total depth. Defining the vertically-integrated velocities (external mode) \((\bar{u}, \bar{v})\), and the fluctuations \((u', v')\) (internal-mode) as what remains when the mean flow is subtracted

\[
\bar{u} = \int_{-1}^{0} u \, d\sigma \quad \text{and} \quad \bar{v} = \int_{-1}^{0} v \, d\sigma
\]

\[
u' = u - \bar{u} \quad \text{and} \quad v' = v - \bar{v}
\]

then the internal-mode velocity equations are derived by subtracting the equations for the external-mode velocity from the equations for the total velocity.

The equations are solved in an Arakawa-C type grid mesh using a finite difference approximation. Variables are placed at different locations within a cell to enhance the numerical solution. For the external-mode equations, where horizontal viscosities are updated, the important variables are the flow variables \(U = \bar{u} H\) and \(V = \bar{v} H\), and the water level, \(h\). The equations used in the model are solved through the semi-implicit numerical scheme of Abbott, using an ADI procedure. For the internal-mode, the strategy for solving the equations is to solve \(u'\) or \(v'\) implicitly by a sweep over the vertical direction. As stated before, decomposition into two modes allow us to utilize a less restrictive time-step than for an equivalent explicit scheme. Levels in the vertical direction extend from the surface at \(\sigma = 0\) to the bottom at \(\sigma = -1\).

The vertical velocity component obtained through the internal mode is a vertical transformed velocity \(\omega\), which physically represents the velocity component normal to the \(\sigma\) surfaces. The transformation to the \(x, y, z\)-system is obtained through the following relation (Mellor, 1998):

\[
w = \omega + u \left( \sigma \frac{\partial h}{\partial x} + \frac{\partial \eta}{\partial x} \right) + v \left( \sigma \frac{\partial h}{\partial y} + \frac{\partial \eta}{\partial y} \right) + \sigma \frac{\partial h}{\partial t} + \frac{\partial \eta}{\partial t}
\]

where \(h(x, y, t) = H(x, y) + \eta(x, y, t)\).

### 2.2 Boundary and initial conditions

The boundary condition at the entrance (lower end) can be of two types; it can be either a water level (tide height) or a flow-rate. In the first case, the water elevation is specified by an algebraic equation of the type \(h = h(x, y, t)\) or through data values specified in a file. In the second case, the water elevation is defined mathematically or specified by points in the vertical plane. The boundary condition at the upper end (outflow condition) is chosen according to the study case; it may also be either a water elevation or a flow-
rate. For the outflow condition based on the radiation of mechanical energy, the water level is computed from the outward boundary normal to the coast as 
\[ h_{\text{side}} = h_0 + \frac{U_\eta}{\sqrt{gh}}, \]
where \( \eta \) is the local outward normal direction and \( \sqrt{gh} \) is the shallow-water gravity wave speed. As boundary conditions are also needed at the bottom and at the surface for the internal-mode, the model establishes these conditions. At the bottom of the water column, we have two possibilities: a non-slip (or velocity matching) condition, \( u = -\bar{u} \) or \( v = -\bar{v} \) at \( \sigma = -1 \) (depending on the direction \( x \) or \( y \)), or the slip (or stress matching) condition, where the stress at the bottom has the form \( \tau_b = \Phi(\bar{u} + u) \) or \( \tau_b = \Phi(\bar{v} + v) \). Also at the surface, the stress matching condition for each direction is \( \tau_s = \Phi(\bar{u} + u) \) or \( \tau_s = \Phi(\bar{v} + v) \). At riverine boundaries, the flow-rate, the flow direction and the internal-mode velocities are required. The internal-mode velocity is set by \( u' = u_{\text{up}} \cos(\pi z/H) \) or \( v' = u_{\text{up}} \sin(\pi z/H) \), where \( u_{\text{up}} \) is a user-specified magnitude. The initial conditions for the run are set in one of two ways, either by data in an initial conditions file or by an internal subprogram. The model allows us to consider some interesting possibilities, such as considering or not the internal-mode (computation \textit{quasi-3D vs. 2DH}) and the non-linear terms of the convective transport. Vertical turbulent viscosity is alternatively computed by a Prandtl type model or by a 1DV of the \textit{K-L} type.

2.3 Rough turbulent boundary layer

Momentum equations (1) and (2) have been obtained by time averaging to the basic equations for the horizontal motion flow, and assuming Boussinesq hypothesis. According to this hypothesis, the turbulent or Reynolds stresses, which represent turbulent velocity fluctuations, are written as deformation velocity functions, corresponding to the mean flow and one parameter, which is a turbulence characteristic – the turbulent viscosity. According to this, the velocity components of the basic equations (equations (1) to (4)) of the model represent time-averaging values, and both the horizontal \( (\nu_u) \) and the vertical viscosity \( (\nu_v) \) values include the molecular viscosity and the corresponding component of the turbulent viscosity.

In contrast to the molecular viscosity \( \nu_0 \), the turbulent viscosity, \( \nu_t \), is not a fluid property, but depends strongly on the state of the turbulence and may vary considerably over the flow field. A turbulence model thus usually has the task of determining the distribution of \( \nu_t \) over the flow field, by relating the turbulence correlations to the averaged dependent variables. The MECCA model assumes that turbulent viscosity is identical in both horizontal directions but different in the vertical direction. Horizontal turbulent exchange coefficients, \( \nu_{L} \), are based on the local horizontal velocity shear and a length scale equal to the grid cell size, \( \Delta L \), so that the momentum exchange coefficient is (Hess, 1989):
where $v_{a0}$ is a coefficient with a nominal value of 0.01, and $C_h$ is a background value (1.0 m²/s).

The simplest models for determining the distribution of the turbulent viscosity ($\nu_t$) over the flow field relate $\nu_t$ directly to the local mean-velocity distribution. These models implicitly assume that the turbulence is dissipated by viscous action at the point where it is generated by shear, which means that there is no transport of turbulence (or of quantities characterizing it) over the flow field. In cases where the state of turbulence at a point is influenced significantly by the turbulence generated somewhere else in the flow (or by that generation previously - history effects), the simple models that disregard turbulence transport are inadequate (ASCE Task Committee on Turbulence Models in Hydraulic Computations, 1988).

The turbulent closure, in the $K-L$ type models, is performed by means of two equations for the turbulent kinetic energy $K$ and for the length scale $L$ of the turbulence. Through turbulent kinetic energy, the neighboring regions are considered in the local distribution turbulence. However, it is the length scale $L$, which characterizes the size of the eddies, that contributes most to the turbulent stresses (Antunes do Carmo, 1995).

In our MECCA model version, the exchange coefficient $\nu_{e}$ is alternatively computed by a mixing-length model of the type $\nu_e = \nu_{a0} + \nu_t (1+10R_i)^{0.5}$, where $\nu_{a0}$ is a user-defined coefficient, $\nu_t = \left[ k_{i} \left( 1 - \frac{z}{H} \right) \right]^{1/2} \sqrt{ \left[ \frac{\partial u}{\partial z} \right]^2 + \left[ \frac{\partial v}{\partial z} \right]^2 }$, and the Richardson number $R_i$ is defined as $R_i = \frac{g}{\rho_o \gamma} \frac{\partial \rho}{\partial z} \left[ \left( \frac{\partial u}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial z} \right)^2 \right]^{-1}$, or by a turbulent 1DV model of the $K-L$ type, which solves the following two equations for the $K$ and $L$ quantities (Berthet, 1996):

$$\frac{\partial K}{\partial t} + u \frac{\partial K}{\partial x} + v \frac{\partial K}{\partial y} = \nu_t \left[ \left( \frac{\partial u}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial z} \right)^2 \right] - \frac{g}{\rho_o \gamma} \frac{\partial \rho}{\partial z} + 0.3 \frac{\partial}{\partial z} \left( \sqrt{2KL} \frac{\partial K}{\partial z} \right) - \frac{K \sqrt{2K}}{4L}$$

$$\frac{\partial L}{\partial t} + u \frac{\partial L}{\partial x} + v \frac{\partial L}{\partial y} = 0.175 \frac{\nu_t}{K} \left[ \left( \frac{\partial u}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial z} \right)^2 \right] L + 0.075 \sqrt{2K} - \frac{0.375 \sqrt{2}}{\sqrt{K}} \left[ \frac{\partial \left( \sqrt{2KL} \right)}{\partial z} \right] + 0.3 \frac{\partial}{\partial z} \left( \sqrt{2KL} \frac{\partial L}{\partial z} \right) + 0.4L \frac{\nu_t}{\rho_o K \gamma} \frac{\partial \rho}{\partial z}$$
The turbulent viscosity is then computed by

\[ \nu_v = \frac{\sqrt{2KL}}{4} \frac{1-16.44\Omega}{(1-19.78\Omega)(1-\Omega)} \]  

where \( \nu_v = \frac{4}{3} \nu_v \frac{1-\Omega}{1-16.44\Omega} \), \( \Omega = \frac{4gL^2}{3\rho_0 q^2 \frac{\partial \rho}{\partial z}} \), and \( q^2 = 2K \), and \( q \) is a characteristic turbulent viscosity.

3 Applications and results

Two case studies have been considered in this study; numerical simulations of a re-circulating flow around a groyne, and the flow around a circular pier in a scour hole. Our numerical results are compared qualitatively with other results found in the literature.

3.1 Groyne

The numerical simulation was run for a rectangular groyne under stationary turbulent-flow conditions. The mean water depth was 10 m, and the mean flow velocity in the undisturbed zone was 0.5 m/s. The groyne was rectangular in shape, 150 m long by 10 m wide (Figure 1). The mesh grid utilized was regular, with \( \Delta L = 10.0 \text{ m} \). At the input boundary, a mean velocity profile of 0.5 m/s, varying from 0.4 m/s at the bottom to 0.6 m/s at the top of the water column, was imposed. A natural (or Newman) condition was considered at the outflow boundary.

![Figure 1: Groyne: Computational domain.](image)

The time-step was \( \Delta t = 0.10 \text{ s} \) and the simulation was performed for a period of 45 minutes. A review of mean velocities indicated that this would be a sufficient simulation time. The horizontal grid mesh involves 50 cells over the channel width, 100 cells in the main flow direction and the vertical is resolved by 20 layers. Simulation required approximately 16 hours on a Pentium III 450 MHz.
10 Hydraulic Information Management

and 64 MB RAM. Figure 2 gives the horizontal velocity field on plane at $z = 0.50$ m above the bed.

Figure 2: Groyne: Modeled velocity distribution close to the bottom (horizontal section at $z = 0.50$ m).

Figure 3 shows $x$-$z$ velocity vectors through a section along the flow, near the groyne base, at $y = 430$ m.

Figure 3: Groyne: Modeled $x$-$z$ velocity distribution through a section along the flow (vertical section at $y = 430$ m).

In general, these results compare well with other numerical results and experimental data described in the literature (see Ouillon and Dartus, 1997).
3.2 Cylindrical pile

The flow around a vertical circular pile was simulated. The flow was performed in a rectangular domain, assuming a pile located in the middle, and with a scour hole around it. A horizontal uniform computational mesh and a vertical non-uniform computational one were constructed on an \((x, y, \sigma)\) coordinate system. The pile diameter was 5.0 m, here approximated by an octagonal section, and the mean water depth was 10 m (Figure 4). At the input boundary, a mean velocity profile of 1.5 m/s, varying from 1.2 m/s at the bottom to 1.8 m/s at the top of the water column was imposed. At the outflow boundary, a natural or Newman condition was considered. The time-step was \(\Delta t = 0.005\) s and the simulation was performed for a period of 45 seconds. As in the previous case, a review of mean velocities indicated that this simulation time was sufficient. The horizontal grid mesh involves 60 cells over the channel width, 120 cells in the main flow direction and 50 layers in the vertical. Simulation required approximately 16 hours on a Pentium III 450 MHz and 64 MB RAM. Figure 4 gives the horizontal velocity field on plane at \(z = 0.20\) m above the bed. Figure 5 shows \(x-z\) velocity vectors through a section at the centerline of the pile, at \(y = 15\) m. Figure 6 shows \(y-z\) velocity vectors through a section downstream of the pile, at \(x = 20\) m.

![Figure 4: Modeled velocity distribution close to the bottom (\(z = 0.20\) m).](image)

![Figure 5: Modeled \(x-z\) velocity distribution through a section at the centerline of the pile (\(y = 15\) m).](image)

![Figure 6: Modeled \(y-z\) velocity distribution through a section downstream of the pile (\(x = 20\) m).](image)
It should be noted that the velocity vectors are not tangent to the wall of the scour hole. The main reason is because the wall is simulated in steps and the flow has an important horizontal velocity component above them. Furthermore, figures (4) to (6) consider the vertical component of the velocity on the \((x, y, \sigma)\) coordinate system, which represents a velocity component normal to the \(\sigma\) surfaces, so the transformation defined by (7) was not applied. Nevertheless, in general, our results compare well with other numerical results and laboratory observations found in the literature (see Richardson and Panchang, 1998).

4 Conclusions

Considering the non-linear terms of the convective transport and the vertical turbulent viscosity computed by the 1DV turbulence model of the \(K-L\) type, as presented in the text, this version of the MECCA model has shown it is able to reproduce the re-circulating flow around a groyne, and also the flow around a pile which has a scour hole around it. We therefore believe that it is a valuable tool for studying the flow around obstacles in coastal and estuarine zones. The simulated results are encouraging enough to proceed with further model developments concerning the sediment transport in these regions.

Acknowledgments

This work has been performed in the scope of the Project PRAXIS/3/3.1/CEG/2503/95, financed by FCT – Portugal.

References


