Validation and calibration of a 3D tidal model of the Hauraki Gulf

C.D. Christian & P.A. Corney

Department of Civil and Resource Engineering, The University of Auckland, New Zealand.

Abstract

A 3D numerical model of the flow in estuaries and coastal seas has been developed. Several options have been included for the turbulence closure. The first part of the paper describes the model and its validation against existing models. The model is then applied to the Hauraki Gulf and the second part of the paper describes this application and the development of a field work program for calibration. The model has been developed as a tool for predicting wind generated surface currents. This will provide a tool for determining water quality issues in a body of water, which is used as a major recreation area.

1 Introduction

The Hauraki Gulf is a shallow sea bounded in the west, south and east by New Zealand’s North Island (see Fig 1). It is the gateway to New Zealand’s largest port, the Port of Auckland, and is used extensively for recreational activity such as fishing, swimming, boating and diving. Water Quality is of great importance for all these activities. The Hauraki Gulf is also the site for the America’s Cup 2000 race series. The venue is exposed to a variety of wind strengths and directions and also experiences moderate tidal flows from a tidal range of 2-3 metres. There is considerable interest from competitors concerning the effect of wind on the tidal currents. The yachts competing in the Cup have a draught of 4-5 metres so are influenced by the surface currents generated by wind generated currents. The initial objective of this model was to determine how differing wind
patterns generate surface currents and if they differ across the race course. In a semi-enclosed sea area with tidal flows this is likely to be of significance to the racing tacticians because as little as 1% performance change can influence the race outcome significantly.

There is also a much more mainstream engineering desire to examine these wind generated surface currents, since the surface currents are responsible for the transport of low density pollution as well as such things as algal bloom.

One of the major tasks when constructing such a model is to firstly validate the model in terms of the solution of the governing equations, and then to calibrate the model against measured conditions. This paper describes the methodology for these two tasks.

2 The hydrodynamic model

The governing system of equations that describe the fluid flow in estuaries and coastal seas are based upon the incompressible Reynolds averaged Navier Stokes equations. In Cartesian co-ordinates, ignoring heat and salt balances, these equations are the continuity equation of mass, and momentum equations for the x, y and z directions. In addition to the above a wind shear stress is added at the water air interface.

The 3D hydrodynamic numerical model being developed is based on the model originally presented by Casulli & Cheng [1]. This model is based on the previous 2D model of Casulli [2]. Unlike many other 3D models the primitive equations are solved without mode splitting thereby negating possible errors involved with the consistency of velocities and boundary conditions between the internal and external modes. The mathematical model is essentially a series of layers for which a set of equations similar to the depth-averaged equations are solved. The governing equations are discretised and solved using a semi-implicit technique such that the stability of the system does not depend upon the wave celerity. For each timestep only the barotropic terms and the vertical eddy viscosity terms are finite differenced implicitly in the momentum equations. The resulting set of equations forms a linear 5 diagonal system whose only unknowns are the surface elevations. Such a system is symmetric and positive definite and so can be solved easily and effectively using a pre conditioned conjugate method. Once found, these elevations or in particular the barotropic pressure gradients can be substituted into the momentum equations to find the fluid velocities at the new time step.

If the convective terms of the momentum equation are solved using a semi implicit Langrangian scheme then the system can be shown to be stable under a mild stability condition (Casulli & Cattani [3]). In this scheme the convective terms for each grid point at each time step are found by tracing back along an advection characteristic curve to the start of the time step.
This scheme also has the advantage that when only one layer is used to model the vertical water column the algorithms reduce to the depth-averaged equations. Flooding and drying can also be easily accounted for. The algorithms are highly vectorisable for efficient use on vector computers.

Space precludes the full presentation of the details of the numerical solution and this will be presented in a further paper.

Several authors have made refinements to this base model including the use of \( \sigma \) co-ordinates (Stansby & Lloyd [4], Li & Zhang [5]), \( k-\varepsilon \) turbulence closure (Stansby [6]) and a second order Crank Nicholson time stepping scheme (Casulli & Cattani [3]).

These have been included here as options. A common problem when using \( \sigma \) co-ordinates is that for steeply sloping bathymetry the horizontal gradients are inaccurate when calculated in the \( \sigma \) frame of reference. To overcome this all horizontal gradients are calculated in real space.

### 2.1 Vertical Eddy Viscosity Distribution

An appropriate model for the turbulence closure is a major part of producing a realistic and stable model. There are a variety of magnitudes and forms for the vertical eddy viscosity distribution, which obviously affects the vertical mixing, which in turn affects, the vertical velocity profile. As part of this project much effort has been invested in developing a realistic representation. Methods tested include, a mixing length model, a parabolic distribution of eddy viscosity. Recently the \( k-\varepsilon \) method has been incorporated into the model.

#### 2.1.1 The Mixing Length Model

The mixing length model may be used in the tidal model. The vertical eddy viscosity is related to the mean velocity gradient by means of a mixing length, \( l_m \). For a given horizontal point the eddy viscosity is given by

\[
\mu = l_m \left[ \frac{\partial u}{\partial y} \right] \tag{1}
\]

The mixing length is constant throughout the depth and has a magnitude of 0.09\( h \), where \( h \) is the total water depth, but at a depth of 0.2\( h \) it reduces linearly to zero at the sea bed.

#### 2.1.2 Parabolic Distribution of Eddy Viscosity

A second alternative for the turbulence closure is the parabolic distribution of eddy viscosity in the form
where $\lambda$ is a fitting constant equal to 0.35, $\tau_s$ is the surface shear stress, $\rho$ is the fluid density and $z_{bh}$ and $z_{sh}$ are surface and bed characteristic lengths normalised to the water depth, $h$. Values of 0.00022 and 0.00014 were used for $z_{bh}$ and $z_{sh}$.

Wu & Tsanis [7] indicate that this form of the vertical eddy viscosity gives a good prediction of the vertical current profile for steady winds in open seas.

### 2.1.3 $k - \varepsilon$ turbulence model

This model attempts to diffuse the turbulence through the model and generate turbulence and is intuitively more realistic than the two previous models.

The local eddy viscosity is given by

$$
\nu_t = C_\mu \frac{k^2}{\varepsilon}
$$

the variables are found by solving

$$
\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + \nu_t \frac{\partial U_i}{\partial x_j} \frac{\partial U_j}{\partial x_i} - \varepsilon
$$

$$
\frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\nu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) + C_1 \nu_t \frac{\varepsilon}{k} \frac{\partial U_i}{\partial x_j} \frac{\partial U_j}{\partial x_i} - C_2 \frac{\varepsilon^2}{k}
$$

The accepted values for coefficients are:

$C_\mu = 0.09$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $\sigma_k = 1.30$ and $\sigma_\varepsilon = 1.30$.

### 3 Test cases

These test cases are used as part of the validation exercise for the model. Having checked the code thoroughly it remains to check the overall performance of the model. Ideally it would be useful to have some full scale 3D flow fields with well defined boundary conditions for comparison. Well documented tests like this are very rare. The authors have identified several papers reporting experimental results, such as – Leschziner & Rodi [8], Koutitas & O’Connor [9], Nakagawa & Nezu [10] and Basara & Younis [11]. These all describe small laboratory models and it is felt that a larger test bed would be more appropriate in order to try and simulate real scale turbulence structures. However the model has been compared
quantitatively with some of these results and generally there is good agreement between model and experiment. Two cases are presented here which illustrate the model behaviour.

3.1 Sharp 90° Bend

This example consists of a rectangular channel 500 m wide and 20 m deep. The channel turns through 90°. The outer length of the channel is 1500 m. A 50 mm water level difference drives a flow at a velocity of approximately 0.5 m/s. The problem uses a constant value for eddy viscosity and run until a steady state result was established. The solutions used 4 layers. The solution is similar to those generated with other codes and clearly shows the flow separation in the lee of the corner.

![Figure 1 Flow through a bend, showing velocity field](image)

3.2 Expansions

The above test is one which could be generated easily with a 2D flow model, so that a test involving 3D flow has been developed which has a sudden expansion combined with a backward facing step. The test section is 500 m long and 200 m wide and 10 m deep at the deepest part. The entry section is 100 m long and 5 m deep. The flow is driven by a 50 mm water level difference. The $k-\varepsilon$ turbulence closure method has been used for this example. The results are shown below.
The Hauraki Gulf is a shallow sea bounded in the west, south and east by New Zealand’s North Island (Figure 3). It is an area of extensive use both recreationally and commercially including fishing, swimming, diving and sailing. It is the gateway to New Zealand’s largest port, the Port of Auckland and is also the site of the next Americas Cup regatta in 2000.

There have been several previous numerical studies of the Hauraki Gulf region (Bowman & Chiswell [12], Grieg & Proctor [13], Proctor & Grieg [14 and Corney & Christian [15]). These studies investigated both the M2 tidal response and the response from steady state winds. All the previous studies were based on two dimensional depth averaged models.

5 Field studies

Since the major aim of this study was to set up a model for the Hauraki Gulf, a series of field studies was deemed necessary to establish boundary conditions and to calibrate the final model. The model is driven by water level boundary conditions along the northern and eastern boundaries. These levels are driven by tidal fluctuations with a phase difference across each one. There are several tide gauges in the gulf and these were used as an initial check on the model accuracy. There was a reasonable agreement between the model and the tide gauges.
Figure 3 Location of Hauraki Gulf showing bathymetry.
Figure 4 Residual surface currents from k-ε model

Figure 5 Residual surface currents from mixing length model
however there are discrepancies between some of the currents recorded on hydrographic charts and the model output. Other authors have noted these discrepancies as well. Thus a series of field studies were instigated for the summer of 1999-2000. The aim has been to determine a series of 3D current profiles over the inner part of the gulf. At the same time it was realised that the currents at the boundaries may not be specified correctly. Anecdotal evidence indicates that some large scale eddying occurs along the eastern boundary with a reverse flow through the boundary at certain tidal stages. Thus part of the program was aimed at providing better boundary conditions.

The field work was based around a current metering program using an acoustic doppler current profiler attached to a 12 m survey vessel. The program was to start in November 1999 and be completed in January 2000. It was intended to survey along the boundaries of the model over a set of large spring tides. The inner surveys have almost been completed however at the time of going to press the boundary studies have been delayed due to bad weather. It is planned to complete these studies over the next few months if the weather is co-operative.

6 Discussion

One of the aims in attempting a calibration of the Hauraki Gulf model was to examine the various turbulence closure models. It was felt that the $k-\varepsilon$ method would provide a better model of the residual currents at slack water, since it conserves the eddy viscosity through regions of low velocity gradient. Other methods, such as mixing length do not do this effectively. An example of this is shown in figures 4 and 5, which show local surface currents in for slack water for a mixing length closure and $k-\varepsilon$ method. Clearly the $k-\varepsilon$ method generates more residual currents and agrees better with the initial field studies.

7 Conclusions

Results have been presented for a 3D free surface flow model which have been used to validate the software. The output from the model has been found to agree with standard cases, both quantitatively and qualitatively. The final calibration of the model in terms of field data has been described and early indications are that there is reasonable agreement with the model. Final calibration requires the completion of the field-work.

8 References


