Flow computations around bridge piers

J.V. Soulis

Fluid Mechanics/Hydraulics Division, Civil Engineering Department, Demokrition University of Thrace, Xanthi, 67100, Hellas, Greece

Abstract

This research work presents the formulation and application of a new mathematical model for the steady, two dimensional, viscous flow in open channels. The numerical technique used is a combination of the finite-element and finite-difference methods. The grid used to represent the physical problem can therefore be curvilinear and irregular, whereas conventional models require regular square grids. The viscous flow stresses are described using fixed value eddy viscosity coefficient. The results are compared with the predictions of a finite-element model which solves the Saint Venant equations on a grid composed of triangular finite-elements. Application regarding flows around circular bridge piers show the comparative performance of the two numerical methods.

1 Introduction

All bridges and structures associated with waterways are potentially at risk from hydraulic action. Therefore, knowledge of the flow field is of predominant importance. A key advantage of non-orthogonal curvilinear grids over regular grids is that lines may coincide exactly with the physical boundaries of the problem, which in turn simplifies the application of the boundary conditions on the solid surfaces. Neilsen and Skovgaard[1] showed that boundary fitted grids can be more accurate than Cartesian grid models for their test cases.

Amongst others, Soulis and Bellos[2] presented two numerical solutions for the computation of two dimensional, inviscid, supercritical, free surface flow including the friction and slope terms. The first numerical solution was obtained by using the well known numerical scheme of MacCormack; the second solution was obtained by solving the flow equations in integral form to a series of finite volumes with adjacent volumes sharing a common face.

Having recognized the importance of a depth averaged model, which was developed by Soulis [3], it was decided to expand it so as to include the viscous flow effects. At the early stages of the computing code developments a constant value eddy viscosity
coefficient was applied and the appropriate stresses were accordingly calculated. The eddy viscosity terms, although approximations of the effective shear terms, have far reaching consequences in that they enable the two dimensional treatment of the predominantly three dimensional features of the flow phenomena, with the computational advances associated with it.

2 Problem definition

The depth-averaged shallow water equations which describe a free surface flow field in Cartesian coordinates are,

\[
\frac{\partial h}{\partial t} + \frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y} = 0
\]

x-momentum equation
\[
\frac{\partial (hu)}{\partial t} + \frac{\partial (gh^2/2+hu^2)}{\partial x} + \frac{\partial (huv)}{\partial y} = -gh(\text{Sfx} - Sfx) + \frac{\partial}{\partial x}\left[v\left(\frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y}\right)\right] + \frac{\partial}{\partial y}\left[v\left(\frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y}\right)\right]
\]

y-momentum equation
\[
\frac{\partial (hv)}{\partial t} + \frac{\partial (gh^2/2+hv^2)}{\partial y} + \frac{\partial (huv)}{\partial x} = -gh(\text{Sfy} - Sfy) + \frac{\partial}{\partial x}\left[v\left(\frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y}\right)\right] + \frac{\partial}{\partial y}\left[v\left(\frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y}\right)\right]
\]

The friction slopes are defined as,
\[
\text{Sfx} = \frac{u(u^2 + v^2)^{1/2}}{hC^2}, \quad \text{Sfy} = \frac{v(u^2 + v^2)^{1/2}}{hC^2}
\]

where C is the Chezy flow friction coefficient. The problem must be closed with the appropriate boundary conditions.

3 Transformation-solution of the shallow water equations

The essence of the proposed numerical scheme is that distorted squares in the physical domain will be separately mapped into squares in the computational domain by independent transformations from Cartesian \((x,y)\) to local \((\xi,\eta)\) coordinates, see Fig. 1. A finite volume which is a quadrilateral in the physical plane or just a square in the computational plane is formed by four nodes (linear element) located at the four corners of the element. The quadrilaterals are packed around the boundaries of the hydraulic structure and cover the whole flow field, see Fig. 2. If \(x_1, y_1, i=1,2,3,4\) are the coordinates of a finite-volume then,
\[
x = x_1N_1 + x_2N_2 + x_3N_3 + x_4N_4, \quad y = y_1N_1 + y_2N_2 + y_3N_3 + y_4N_4
\]

\[
N_1 = \frac{(1-\xi)(1-\eta)}{4}, \quad N_2 = \frac{(1+\xi)(1-\eta)}{4}, \quad N_3 = \frac{(1+\xi)(1+\eta)}{4}, \quad N_4 = \frac{(1-\xi)(1+\eta)}{4}
\]

Let \([J^-1]\) be the transformation matrix from the physical to the local coordinate system,
\[
[J^-1] = \begin{vmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta}
\end{vmatrix}
\]
The following equations hold Soulis[3],
\[
\frac{\partial x}{\partial \xi} = J^{-1} \frac{\partial \eta}{\partial y}, \quad \frac{\partial x}{\partial \eta} = -J^{-1} \frac{\partial \xi}{\partial y}, \quad \frac{\partial y}{\partial \xi} = -J^{-1} \frac{\partial \eta}{\partial x}, \quad \frac{\partial y}{\partial \eta} = J^{-1} \frac{\partial \xi}{\partial x} \tag{8}
\]
where $J^{-1}$ is the determinant of matrix $[J^{-1}]$. Under the aforementioned transformation of Eqs. (1)-(3) into the local coordinate system they assume the form Soulis et al [4],

continuity equation
\[
\frac{\partial (J^{-1} h)}{\partial t} + \frac{\partial (J^{-1} h U)}{\partial \xi} + \frac{\partial (J^{-1} h V)}{\partial \eta} = 0 \tag{9}
\]
\[\xi\text{-momentum equation}\]
\[
\frac{\partial (J^{-1} h u)}{\partial t} + \frac{\partial (J^{-1} h U u + \frac{\partial \xi}{\partial x} g h^2 / 2)}{\partial \xi} + \frac{\partial (J^{-1} h V)}{\partial \eta} = J^{-1} g h (\text{Sox} - \text{Sfx}) + \frac{\partial}{\partial \eta} \left[ J^{-1} \left\{ \frac{\partial (2 h U)}{\partial \eta} \right\} \right]
\]

\[\eta\text{-momentum equation}\]
\[
\frac{\partial (J^{-1} h v)}{\partial t} + \frac{\partial (J^{-1} h V v + \frac{\partial \xi}{\partial y} g h^2 / 2)}{\partial \xi} + \frac{\partial (J^{-1} h U)}{\partial \eta} = J^{-1} g h (\text{Soy} - \text{Sfy}) + \frac{\partial}{\partial \eta} \left[ J^{-1} \left\{ \frac{\partial (2 h V)}{\partial \eta} \right\} \right]
\]
where $U, V$ are the velocity components along the $\xi, \eta$ directions respectively. For a control volume of unit height and for a given time step $\Delta t$ the PDE (9)-(11) may be written as,
\[
-\Delta (J^{-1} h) = [\Delta (J^{-1} h U) \Delta \eta + \Delta (J^{-1} h V) \Delta \xi] \frac{\Delta t}{\Delta \xi \Delta \eta} \tag{12}
\]
\[
-\Delta (J^{-1} h u) = \{\Delta [J^{-1} \{ h U U + \frac{\partial \xi}{\partial x} g h^2 / 2 \} \frac{\partial (h U)}{\partial x} + \frac{\partial (h V)}{\partial y} \frac{\partial (h V)}{\partial x}] \} \Delta \eta + \Delta [J^{-1} \{ h V v + \frac{\partial \xi}{\partial y} g h^2 / 2 \} \frac{\partial (h V)}{\partial x} + \frac{\partial (h U)}{\partial x} \frac{\partial (h U)}{\partial y}] \} \Delta \xi \frac{\Delta t}{\Delta \xi \Delta \eta} + \]

\[
J^{-1} g h (\text{Sox} - \text{Sfx}) \frac{\Delta t}{\Delta \xi \Delta \eta} \tag{13}
\]
\[
-\Delta (J^{-1} h v) = \{\Delta [J^{-1} \{ h V v + \frac{\partial \xi}{\partial x} g h^2 / 2 \} \frac{\partial (h V)}{\partial x} + \frac{\partial (h U)}{\partial x} \frac{\partial (h U)}{\partial y}] \} \Delta \eta + \Delta [J^{-1} \{ h U U + \frac{\partial \xi}{\partial y} g h^2 / 2 \} \frac{\partial (h U)}{\partial x} + \frac{\partial (h V)}{\partial x} \frac{\partial (h V)}{\partial y}] \} \Delta \xi \frac{\Delta t}{\Delta \xi \Delta \eta} + \]

\[
J^{-1} g h (\text{Soy} - \text{Sfy}) \frac{\Delta t}{\Delta \xi \Delta \eta} \tag{14}
\]

Figure 3 also shows the notation used for the flux balancing across a finite-volume. An XFLUX and a YFLUX at point $i, j$ are defined as,
\[
\text{XFLUX}_{i,j} = [(J^{-1} h U)_{i+1,j} + (J^{-1} h U)_{i,j}] \frac{\Delta \eta}{\Delta \xi} \tag{15}
\]
\[
\text{YFLUX}_{i,j} = [(J^{-1} h V)_{i+1,j} + (J^{-1} h V)_{i,j}] \frac{\Delta \xi}{\Delta \eta} \tag{16}
\]
with similar notation for the $\xi$-momentum and $\eta$-momentum fluxes. The terms $\Delta (J^{-1} h U)$ and $\Delta (J^{-1} h V)$ of the RHS of Eq.(12) are defined as,
\[
\Delta (J^{-1} h U) = (\text{XFLUX})_{i+1,j} - (\text{XFLUX})_{i,j-1} \tag{17}
\]
Figure 1 Distorted squares mapped into squares

Figure 2 Computational grid used for bridge circular piers. Flow is from the left. All dimensions in meters. A 42x74(=3108) computational grid is shown.

Figure 3 Flux balancing across a finite volume

Figure 4 Solid boundary finite-volume for the first computational row (i=1, j=1, JM). The velocity is tangential to the boundary face
Figure 9 TELEMAC-2D predicted contours of equal water depth (m) for $h_2=0.0965$ (m), $u_1=0.177$ (m/s), $C=14.6$ and $v=0.001$ (m²/s)

Figure 10 TELEMAC-2D predicted contours of equal Froude number for $h_2=0.0965$ (m), $u_1=0.177$ (m/s), $C=14.6$ and $v=0.001$ (m²/s)

Figure 11 TELEMAC-2D predicted contours of equal $u$ (m/s) for $h_2=0.0965$ (m), $u_1=0.177$ (m/s), $C=14.6$ and $v=0.001$ (m²/s)

Figure 12 TELEMAC-2D predicted contours of equal $v$ (m/s) for $h_2=0.0965$ (m), $u_1=0.177$ (m/s), $C=14.6$ and $v=0.001$ (m²/s)