第33回水理講演会論文集　1989年２月

Flow in a river confluence with a levee predicted by three-dimensional model

Nobuyuki Tamai, Professor
S. B. Weerakoon, Grad. Student
Dept. of Civil Engg., University of Tokyo.

A three-dimensional computational model, based on finite volume method and k-ε turbulence model, for predicting steady state confluence flow is presented. The confluence region is covered by a curvilinear grid which is generated numerically using a quasiconformal mapping method. The computation is done in the transformed domain which is a parallelipiped. The computed results agree well with the experimental results. The performance of a levee which facilitates two streams to mix gradually and thereby to reduce the superelevation and the strong secondary flow in the main stream, is demonstrated.
Keywords : confluence, grid generation , levee, secondary flow, three-dimensional flow computation

1. Introduction

The confluences, where the tributaries join with the main streams are common in river systems. The flow situation in a confluence is very complex in three dimensions with the secondary flow, separation and strong mixing of two streams. It has been understood that the strength of those phenomena depend mainly on flow ratio, confluence angle and Froude number(Mosely). The levee is thus constructed in a confluence in order to reduce the effective confluence angle and thereby to facilitate gradual mixing. Three-dimensional model for predicting the flow in a confluence with confluence angle of 30 degrees is given by Tamai & Ueda. However the model is restricted for small confluence angles due to the restraints in the grid system adopted in the computation. On the other hand, the role of a levee is more important when the confluence angle is large. Therefore, this paper describes a three-dimensional computational model which employs numerical grid generation technique and hence capable of predicting the flow in an arbitrary confluence. The model is based on the finite difference solution of the time-averaged Navier Stokes equations, continuity equation and the standard k-ε model. The streamwise diffusion is neglected in the model as partially parabolic flow computation procedure is adopted.

2. Grid Generation

A quasiconformal mapping method, in which the boundary correspondence is given and remained unchanged and thus coordinate lines can be distributed at required spacings, is employed to generate the smooth and closely orthogonal curvilinear grids. Upon the condition that bed level variation is smooth and not large we can generate the grid in two-dimensional flat surface and then can be extended to the third dimension by the interpolation according to the required spacings. If y^1, y^2 and x^1, x^2 are the Cartesian and curvilinear coordinates respectively, then starting from the orthogonal relations,

$$\left(\frac{\partial y^2}{\partial x^1}\right) = D\left(\frac{\partial y^1}{\partial x^2}\right) \tag{1}$$

$$\left(\frac{\partial y^1}{\partial x^1}\right) = -D\left(\frac{\partial y^2}{\partial x^2}\right) \tag{2}$$

following two equations can be derived.

$$\frac{\partial}{\partial x^1}\left(\frac{1}{D}\frac{\partial y^1}{\partial x^1}\right)+\frac{\partial}{\partial x^2}\left(D\frac{\partial y^1}{\partial x^2}\right) = 0 \tag{3}$$

$$\frac{\partial}{\partial x^1}\left(\frac{1}{D}\frac{\partial y^2}{\partial x^1}\right)+\frac{\partial}{\partial x^2}\left(D\frac{\partial y^2}{\partial x^2}\right) = 0 \tag{4}$$

where D is the aspect ratio given by $D = \sqrt{\frac{g_{11}}{g_{22}}}$

With D explicitly given through an initial guess for y^1, y^2, the solution to the elliptic system of equations (3) and (4) under the specified spacings of grids at the boundaries as boundary conditions maps the square meshes in the transformed region onto the curvilinear meshes in the physical region. D is computed from new coordinates of and smoothed over the whole region. The smooth and closely orthogonal grid is then obtained by repeating the sweeps for the solution several times with new values as initial guess for y^1, y^2.. Part of grid obtained in this method is shown in figure(2).

The transformation relations such as metric tensor, the Christoffel symbols are computed numerically in order to transfer the three-dimensional confluence region with curvilinear meshes to a parallelipiped with square meshes.

3. Governing equations and turbulence model

For incompressible flows, continuity equation and time-averaged Navier Stokes equations applicable in nonorthogonal curvilinear coordinates can be written in the following tensor form using contravariant velocity components.

$$U^i_{;i}=0 \tag{5}$$

$$(U^iU^j)_{;j} = f^i-\frac{1}{\rho}p^i_{,}+(\nu U^{ij}_{;}-\overline{u^iu^j})_{;j} \tag{6}$$

where U^i = contravariant component of mean velocity, u^i = contravariant component of turbulent velocity, ; = covariant derivative, p = piezometric pressure and ν = kinematic viscosity. Eddy-viscosity concept and the standard k-ε model are used to compute the turbulent stresses. The related equations can be written as,

$$-\overline{u^iu^j} = \nu_t\left(U^i{}_{;}{}^j+U^j{}_{;}{}^i\right)-\frac{2}{3}g^{ij}k$$

$$\left(kU^j\right)_{;j} = \left(\frac{\nu_t}{\sigma_k}k_{,}{}^j\right)_{;j}+\nu_t\left(U_{i;}{}^j+U^j_{;i}\right)U^i_{;j}-\varepsilon \tag{7}$$

$$\left(\varepsilon U^j\right)_{;j} = \left(\frac{\nu_t}{\sigma_\varepsilon}\varepsilon_{,}{}^j\right)_{;j}+c_1\frac{\varepsilon}{k}\nu_t\left(U_{i;}{}^j+U^j_{;i}\right)U^i_{;j}-c_2\frac{\varepsilon^2}{k} \tag{8}$$

$$\nu_t = c_\mu\frac{k^2}{\varepsilon} \tag{9}$$

$$c_\mu=0.09,\quad c_1=1.44,\quad c_2=1.92,\quad \sigma_k=1.0,\quad \sigma_\varepsilon=1.3$$

where k = turbulent kinetic energy, ε = dissipation rate and ν_t=eddy viscosity.

4. Solution procedure

The equations (5) to (9) are written in the conservative form and integrated over the finite volumes in the transformed plane to form the close set of descrete equations. The hybrid scheme is used to evaluate the coefficients in the cross-stream and is assumed to be numerically accurate because the velocity vectors are closely in line with the numerical grid lines. The partially parabolic flow assumption is made here in order that downstream marching solution is possible. The pressure correction equation was derived from equation (5) while considering the the effect of all three pressure gradients, risen due to the nonorthogonality of the grids, on the velocity correction terms. A procedure similar to the SIMPLE algorithm is then used to obtain the converge solution by making sweeps from upstream to downstream. Furthermore, wall function method is employed as boundary condition near the wall. The water surface is considered as a symmetrical rigid lid in the computation thus water surface elevation difference is considerd only indirectly by the pressure gradient terms.

Before employing to present complex flow situation, the performance of the model was checked by applying to simple confluence flows(Tamai and Weerakoon).

5.Results and discussion

The model is applied to predict the three-dimensional flow in the confluence of 60 degrees, having rectangular cross sections and a training levee studied experimentally by Tada. The configuration of the confluence is illustrated in figure(1), The side walls of the channels were nominally smooth. The discharge in the main channel was 1.646 l/s and that of branch was 1.366 l/s. The down stream controll and the upstream feeders of the channels were located so that their effect on the confluence is neglegible.

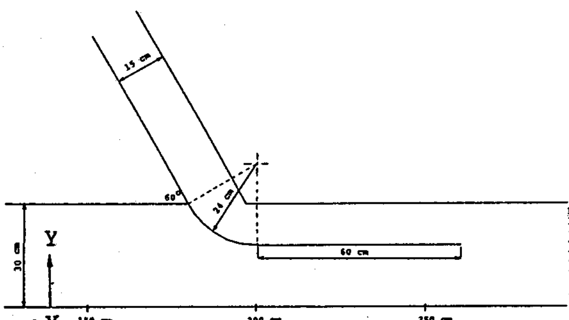


Fig.1 Configuration in experiment

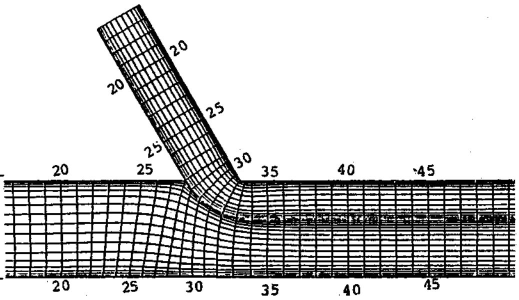


Fig.2 Part of grid

A total length of 450cm while 300cm being after the confluence was considered in the flow computation. The fully developed flows were input at the upstream and the flow was assumed to be developed at the downstream as boundary conditions. The comparison of the longitudinal component of the velocity vector drawn in streamwise vertical sections is shown in figure(3), where h is the depthwise coordinate measured from the bottom and X,Y are measured as indicated in figure(2). The velocity scales are the same for each curve and the 9 curves given in each figure corresponding to the different values of X progressing downstream as given with the figure caption. As can be seen from figure(3), the prediction agrees quite well with the measurments, except the very low velocities appear near the downstream corner of the geometric junction. However, this is the location where the flow separation is very likely to occur at the confluence in which case the model cannot be expected to perform well due to the fact that the model is based on partially parabolic flow assumption. The absence of flow separation in this case can be thought mainly due to the presence of the levee. Also, a small over estimation of the velocities is shown in general throughout this developing flow region. This discrepancy rises to ensure the continuity of the flow, because the actual water depths in this region are greater than the imposed constant water depth corresponding to the actual downstream water depth. Although this is a consequence of the symmetrical boundary condition imposed at the top surface the actual water depths can be calculated through the pressure gradient(Tamai and Ueda). The comparison of water depth variation with experimentally obtained values show satisfactory agreement as depicted

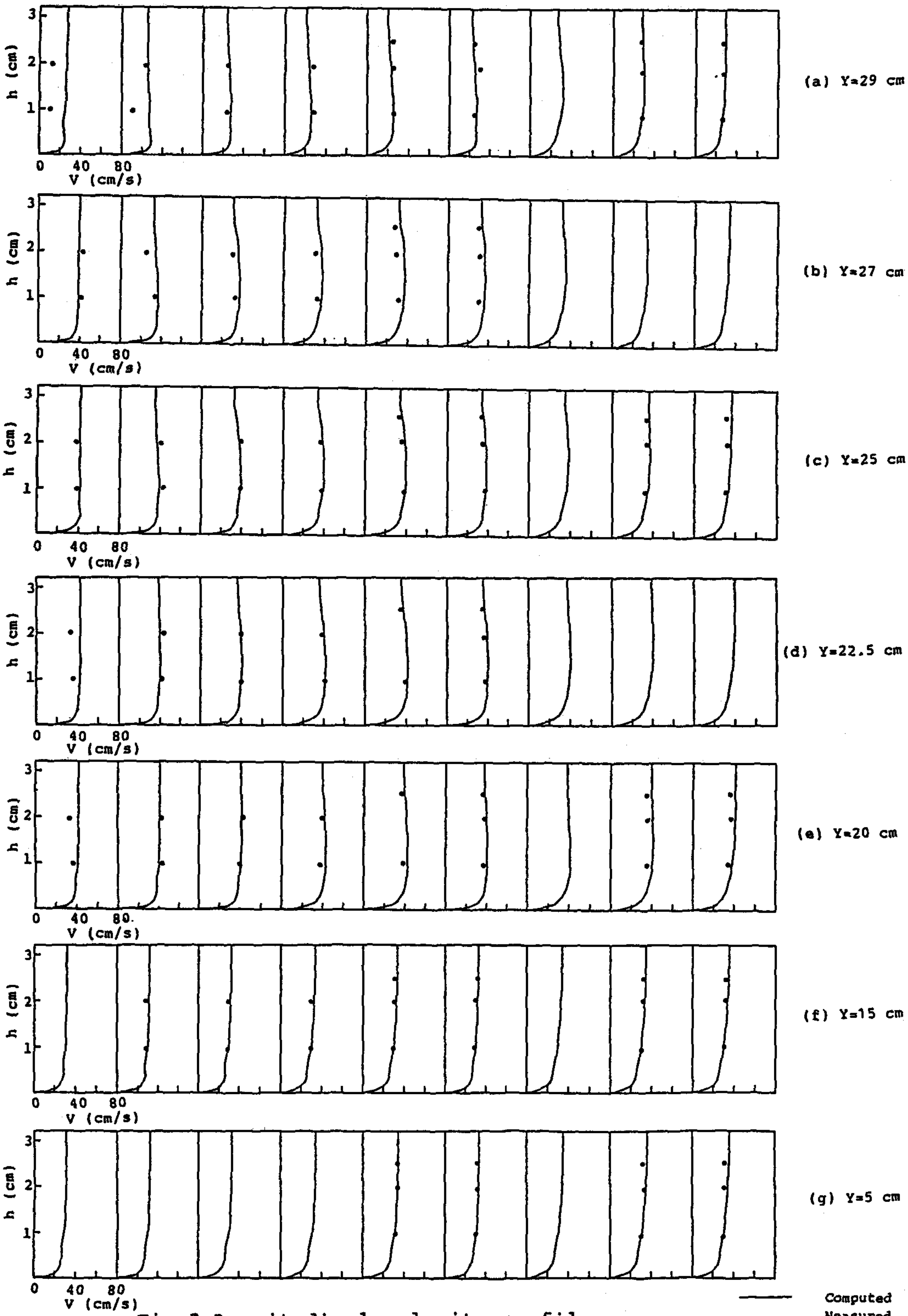


Fig.3 Longitudinal velocity profiles at X values of 200,215,235,265,270,280,325,380,480 cm

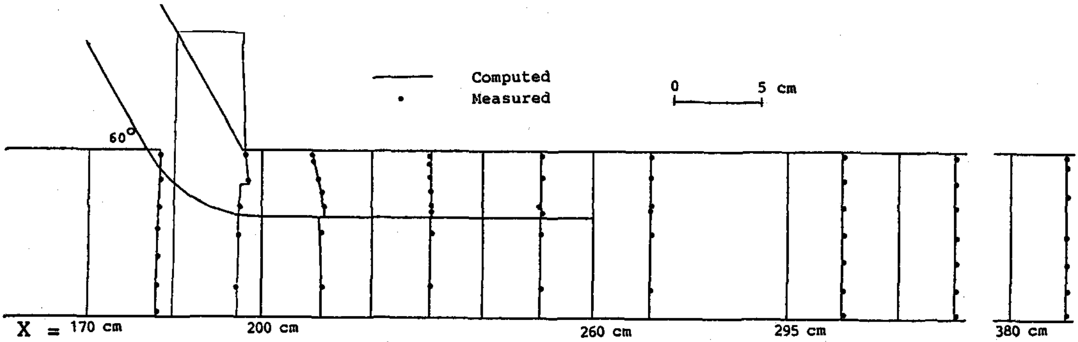


Fig.4 Flow depth variation

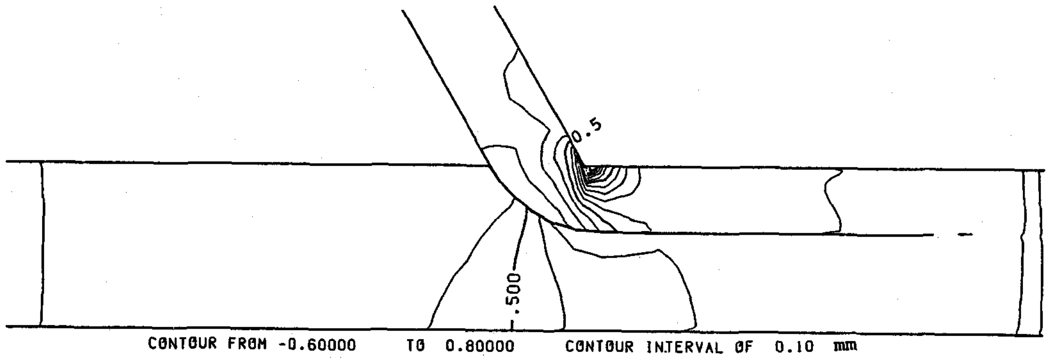
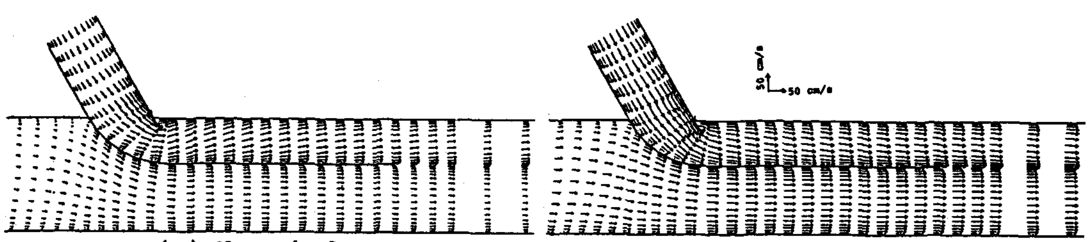


Fig.5 Piezometric pressure variation



(a) Near bed (b) Near surface

Fig.6 Velocity vectors in plane sections

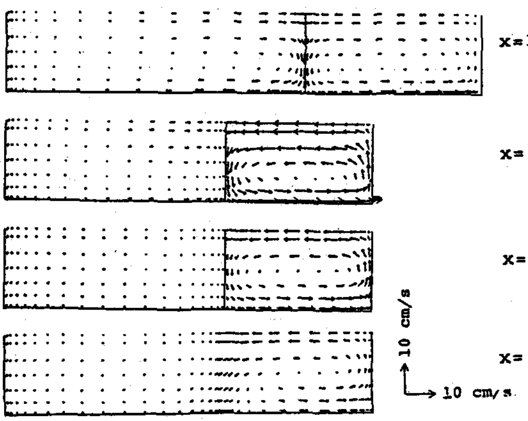


Fig.7 Secondary flow

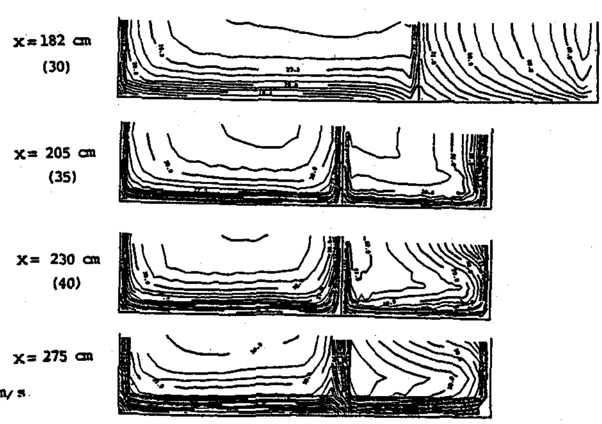


Fig.8 Velocity isovels

in figure(4). Figure(5) shows piezometric pressure contours which correspond to the water surface level variation in the region. According to these figures, the superelevation of the both flow channels towards the levee is prominent. If the levee is absent, strong mixing will occur between two flows and thus the superelevation in the branch channel which is substantial at the present case can be expected to be smaller, but on the other hand the superelevation at the main channel towards the right bank can be expected to be larger. Figure(6) shows the velocity vectors in plane sections at near bottom and near surface. Figure (7) shows the secondary flow vectors in lateral sections. The sections are as shown in the figure(2). The spiral motions which counter rotates appear in two channels at the bothsides of the levee. These two vortexes appear for the reason of continuity as the acceleration and decelaration of the fluids occur in their curved paths. The vortex in the branch channel which is comparatively large is shown to has not decayed completely before two flows join at the end of levee. The presence of levee has blocked the strong mixing of the flows at the junction and thus reduced the secondary flow in the main channel. On the other hand it has blocked momentum transfer thus resulting a slow decaying strong vortex in the branch channel. The training levee has made the mixing of two flows gradual as flows departs the levee thus considerably reduced the superelevation in the main channel appears in confluences without levees (Tamai and Ueda). The figure(8) shows the velocity isovels of the sections. The strong secondary motion has caused the depression of the maximum velocity below the water surface as seen.

6.Conclusions

a). A computational model based on the standard k-ε turbulence model has been presented for the confluences with no separation. The agreement of the computed results with experimental results is generally good.

b). The grid generation method adopted can be used to tackle an arbitrary confluence and is flexible in spacing of grids.

c). The performance of a levee has been demonstrated. The levee prevent strong mixing and facilitates two streams to mix gradually. Thereby superelevation in the junction and the strong secondary flow in the main stream are considerably reduced.

Aknowledgements

The authors are grateful to H.Tada for providing experimental data.

References

1). Demuren, A.O. and W. Rodi : Side discharges into open channels: Mathematical model, J. of Hyd. Engg., ASCE, vol.109, No 2, pp. 1707-1721, 1983.

2). Mosely, M. P. : An experimental study of channel confluences, J. of Geology, Vol.94, pp.535-562, 1976.

3). Patankar, S. V. : Numerical Heat Transfer and Fluid Flow, Hemisphere, N.Y., 1980.

4). Rodi, W. : Turbulence Models and their Applications in Hydraulics, Monograph, IAHR, Delft, The netherlands, 1980.

5). Tada, H. : Experimental report, Osaka College of Technology, 1987

6). Tamai, N. and S. Ueda : Prediction of flow behavior at river confluences by the k-e model, Proc. of the 31st Japanese conf. on Hydraulics, pp.437-442, 1987.

7). Tamai, N. and S. B. Weerakoon : Three-dimensional confluence flow computation using numerically generated grid, Proc. of the 43rd annual conf. of JSCE, pp. 442-443, 1988.

8). Thompson, J. F. and Z. V. A. Warsi : Boundary-fitted coordinate systems for numerical solution of partial-differential equations- A review, J. fo Comp. Physics, 47, pp. 1-108, 1982.