SIMULATION OF SHALLOW FLOWS IN A CIRCULAR BASIN USING COLLOCATED MULTI-BLOCK METHOD

MOHAMMAD REZA HADIAN[1] and AMIR REZA ZARRATI[2]

[1]PhD Student, Department of Civil and Environmental Engineering,
Amirkabir University of Technology, No.424, Hafez Ave., 15914, Tehran, Iran
(Tel: +98-21-64543078, Fax: +98-21-6414213, e-mail: hadian@aut.ac.ir)
[2]Associate Professor, Department of Civil and Environmental Engineering,
Amirkabir University of Technology, No.424, Hafez Ave., 15914, Tehran, Iran
(Tel: +98-21-64543002, Fax: +98-21-6414213, e-mail: zarrati@aut.ac.ir)

Numerical solution of the equations governing the fluid flow has been studied for many years by different researchers. In many cases in nature such as in rivers or coastal currents with small depth to width ratio, shallow water equations could be used to simulate flow conditions. Numerical solution of 2D shallow water equations has been performed for more than 30 years from the early works of Kuipers and Vreugdenhil (1973). Since then several other research works have also been published.

One of the problems in Computational Fluid Dynamics (CFD) is the complexity of flow domain. Using curvilinear coordinates is one of the popular methods to deal with complex geometries. In this method the equations are transformed from Cartesian to boundary fitted coordinate system and are solved in the computational mesh. However, generating appropriate mesh in very complex flow domains is very difficult. For instance, cases with sharp changes in flow boundaries leads to a very poor numerical mesh, considering the smoothness and mesh skewness. This results in inaccuracy and problems in convergence for some of CFD techniques (Wijbenga 1985; Peric 1990, Zarrati et al. 2005).

Among different methods which can be employed to overcome this difficulty, Multi-Block (MB) technique is the more general one that if is combined with curvilinear coordinate system will become a powerful method to model very complex flow boundaries. In this technique the flow domain is decomposed to some sub-domains (blocks) and the equations are solved in every block separately.

In the present work the MB method is used in conjunction with curvilinear coordinate system to tackle with a complex geometry, which is a circular basin with inlet and outlet channels. Without MB method, mesh generation in this case results in very skewed or even triangular control volumes at some regions, which this may cause divergence and/or inaccuracy. Using MB technique, very smooth mesh with suitable size and aspect ratio is generated in the flow domain.

The present model is based on collocated grid arrangement and the control volume method is used for solution of equations. A SIMPLEC-like algorithm is employed to find the water surface elevation. Flow in the circular basin was simulated using 13 blocks and computed flow pattern is shown in Fig. 1. Comparison of results with other numerical models showed the accuracy of the present scheme.

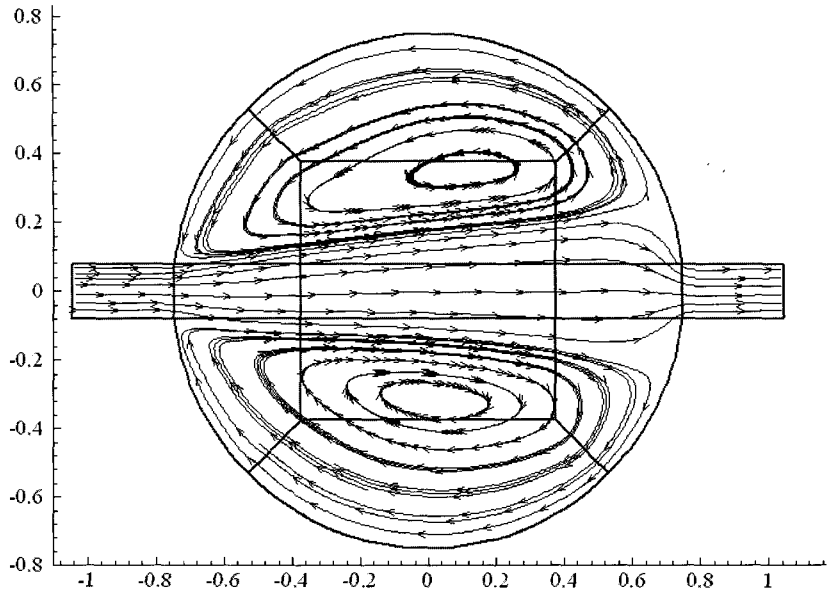

Fig. 1 Flow pattern in the circular basin by present model.

REFERENCES

Kuipers, J. and Vreugdenhil, C.B. (1973). *Calculation of Two-Dimensional Horizontal Flow*, Rep. S163, Part 1, Delft Hydraulics Lab., Delft, Netherlands.

Peric, M. (1990). "Analysis of pressure-velocity coupling on non-orthogonal grids," *Numerical Heat Transfer, Part B*, Vol.17, pp.63-82.

Wijbenga, J.H.A. (1985). "Determination of flow patterns in rivers with curvilinear coordinates," *Proc. 21th IAHR Congress*, Melbourne, Australia, pp.132-139.

Zarrati, A.R., Tamai, N. and Jin, Y.C. (2005). "Mathematical modeling of meandering channels with a generalized depth averaged model," *Journal of Hydraulic Engineering*, ASCE, Vol.131, No.6, pp.467-475.