8th. World Congress on Computational Mechanics (WCCM8)
5th European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 –July 5, 2008
Venice, Italy

A SECOND ORDER METHOD FOR THE RESOLUTION OF THE SHALLOW WATER EQUATIONS WITH TURBULENT TERM

*** J. Fe[1], L. Cueto-Felgueroso[2] and F. Navarrina[3]**

[1] Grupo de Métodos Numéricos en Ingeniería, Universidad de A Coruña, Campus de Elviña, 15071, A Coruña, Spain. jfe@udc.es http://caminos.udc.es/gmni

[2] Aerospace Computational Design Laboratory, Massachusetts Institute of Technology, 77 Massachusetts Ave, Cambridge, MA 02139 USA lcueto@mit.edu

[3] Grupo de Métodos Numéricos en Ingeniería, Universidad de A Coruña, Campus de Elviña, 15071, A Coruña, Spain. fnavarrina@udc.es http://caminos.udc.es/gmni

Key Words: *Finite volumes, upwind, numerical viscosity reduction, turbulence, gradient mean values.*

ABSTRACT

The behaviour of a fluid in 3D may be described by the Navier-Stokes equations which are a hyperbolic system of non linear conservation laws. Their complexity has led to the development of the two-dimensional shallow water equations (2D-SWE) based on some simplifying hypothesis, the most important of which is the hydrostatic pressure distribution [1]. This set of equations describes remarkably well the fluid behavior when the ratio of the depth to the horizontal dimensions is small and the magnitude of the vertical velocity component is much smaller than the magnitude of the horizontal velocity components at the space and time scales of interest for the resolution of a given problem. This situation can be found, for instance, in the flow in channels and rivers or tidal flows.

2D-SWE take into account the effects of turbulence both through the frictional terms and the diffusion-like term, that involves second derivatives. Frictional terms quantify the turbulence effects in the vertical, while the second derivatives term quantifies the turbulent losses produced by the horizontal mixing of momentum. This last term may not be significant in many practical problems when we only need an estimate of energy losses and 2D-SWE are frequently used without considering the second derivatives term, what is a reasonable simplification in many cases but not always. Thus, in the simulation of flows in which recirculation zones play a significant role, the inclusion of this turbulent term may become very important.

The 2D study of viscous fluids with Reynolds numbers below 10.000 can be very accurately solved by the utilization of constant values for the viscosity in the turbulent term [2], but in most cases of practical interest it is neccesary to calculate the turbulent viscosity at every point and a turbulence model is therefore needed. The depth-averaged $k - \varepsilon$ model has been used to obtain the turbulent viscosity by many researchers [3] and it has been chosed for our work.

To discretize the equations, the finite volume method has been used. An important point when working with this method is to properly calculate the numerical flux of the convective term at the cell edges. The

upwinding of the flux term has proved to be a useful technique [4] with the drawback of producing a certain amount of numerical viscosity (or diffusion), which in some cases may be of similar magnitude to the turbulent viscosity. In order to reduce the numerical diffusion the use of an upwinding coefficient in a first order scheme has been described in an earlier work and it gives reasonably accurate results [5]. However, this method reduces also the stability of the numerical process. For this reason a second order method has been developped, which makes use of the mean gradient of the variables in a cell. This mean gradient is calculated from the values at the cell edges, which depend on the values of the variables at the adjacent cells. Thus the considered cell and the ones that surround it are involved.

In order to compare the first and second order methods, the Cavity Flow problem has been used. Calculations have been made for three viscosity values, corresponding to Reynolds numbers of 100, 1000 and 10000. The computational mesh is regular with 81 x 81 nodes. In Figure 1 we represent the resulting streamlines obtained by using the second order method.

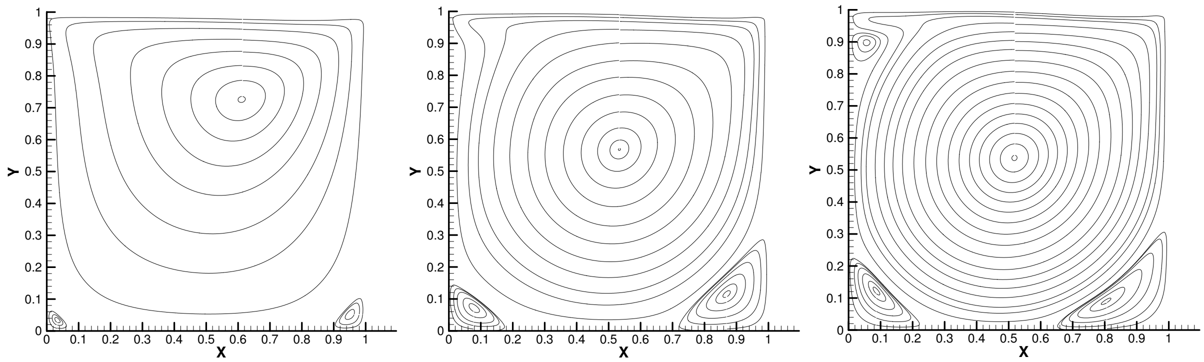

Figure 1: Cavity Flow. Streamlines for different viscosity and Reynolds number values. From left to right: a) $\nu = 0.01$ $(Re = 100)$, b) $\nu = 0.001$ $(Re = 1000)$, c) $\nu = 0.0001$ $(Re = 10000)$.

In the present work: 1) we describe the proposed second order method to discretize the hydrodynamic equations; 2) we compare the first and second order hydrodynamic models, in the Cavity Flow problem, for uniform values of the viscosity and we show the better results obtained with the second order model; 3) we combine the second order hydrodynamic model with the first order $k - \varepsilon$ model, comparing the obtained results with experimental measures.

REFERENCES

[1] C. B. Vreugdenhil. *Numerical methods for shallow-water flow*, Kluwer Academic Publishers, Dordrecht, The Netherlands, 1994.

[2] P. Vellando, J. Puertas and I. Colominas, "SUPG stabilized finite element resolution of the Navier-Stokes equations. Applications to water treatment engineering". *Comput. Methods Appl. Mech. Engrg.*, Vol. **191**, 5899–5922, 2002.

[3] W. Rodi. *Turbulence models and their application in hidraulics. A state-of-the-art review*, IAHR monograph, 3rd. Edition, Balkema, Rotterdam, 1993.

[4] A. Bermúdez, A. Dervieux, J. Desideri and M.E. Vázquez. "Upwind schemes for the two dimensional shallow water equations with variable depth using unstructured meshes". *Comput. Methods Appl. Mech. Engrg.*, Vol. **155**, 49–72, 1998.

[5] J. Fe, L. Cueto-Felgueroso, F. Navarrina and J. Puertas. "Numerical viscosity reduction in the resolution of the shallow water equations with turbulent term". *Int. J. Numer. Meth. Fluids*, (in Press).