Mecánica Computacional Vol. XXI, pp. 2311–2324
S. R. Idelsohn, V. E. Sonzogni and A. Cardona (Eds.)
Santa Fe–Paraná, Argentina, October 2002

BENCHMARK SOLUTIONS FOR THE ASSESSMENT OF COMPUTER CODES BASED ON THE SHALLOW WATER EQUATIONS

Pablo A. Tassi†, and Carlos A. Vionnet†

†Facultad de Ingeniería y Ciencias Hídricas (FICH)
Universidad Nacional del Litoral, CONICET
Ciudad Universitaria, Paraje El Pozo, 3000 Santa Fe, Argentina
e-mail: ptassi@fich1.unl.edu.ar - vionnet@fich1.unl.edu.ar

Key Words: Shallow water, Finite elements, River flow, Analytical solutions.

Abstract.

Several finite element codes for solving the two-dimensional shallow water model are currently available for engineering applications. While most of these codes are capable of computing the water surface elevation and the depth-averaged velocity of free-surface flows in any given geometry, the computing process itself is sometimes hampered with unpleasant consequences for the inexperience modeler. Some of these unwelcome problems may be triggered by the algorithms built into the codes (i.e., depending on the method used to incorporate the required amount of 'upwinding' to cope with the convective terms), or by the uncertainty present in the values of the model parameters that may influence the flow physics. In this communication, the performance of some codes is tested. To that aim, we shall present the discussion of few benchmark solutions that can be used to measure the performance of any computational code based on the shallow water equations. Some of them correspond to rather unrealistic situations that have the purpose of illustrating the capabilities and/or the accuracy/inaccuracy achieved by the codes being tested. Other benchmark solution is related to a real engineering problem that is ambiguously modeled with some of the shallow water codes.

1 INTRODUCTION

The Shallow Water Equations (SWE) are typically used to model river and lake hydrodynamics, floodplain flows, estuarine and coastal circulation as well as long wave runup among many other problems of interest within the engineering community.[1] Indeed, the range of application of the shallow water approximation exceeds by far the scope of the hydraulic engineer. Nowadays, the trend in geomorphology and hydrology is to adopt Computational Fluid Dynamics (CFD) techniques developed by engineers and scientists to study a variety of problems that can be approached with the long wave approximation or the SWE.[2,3] Then, the development of user-friendly interfaces has been so rapid lately that the numerical simulation of complex hydrodynamic problems is today a standard computational task. However, when using a computational code, the engineer somehow surrenders control of the accuracy of the computation to the model developer. In other words, while most of these codes are capable of computing the water surface elevation and the depth-averaged velocity of free-surface flows in any given geometry, the computing process itself is sometimes hampered with unpleasant consequences for the inexperience modeler. Some of these unwelcome problems may be triggered by the algorithms built into the codes (i.e., depending on the method used to incorporate the required amount of 'upwinding' to cope with the convective terms), or by the uncertainty present in the values of the model parameters that may influence the flow physics. Thus, it is still of paramount importance to check the quality of computer codes by comparing its results against standard benchmark cases and real situations as well. In the former case, the code is mostly tested against analytical solutions of the governing equations to check its numerical ability to yield approximate solutions of the proposed governing equations. In the other case, the availability of good quality field data gives the user the possibility to assess the capability of the codes to predict the flow behavior in cases of practical interest.

In this communication, the performance of the RMA-2,[4] the FESWMS-2DH[5] and the TELEMAC-2D[6] shallow water model codes is tested. To that aim, the discussion of several benchmark solutions that can be used to measure the performance of any computational code based on the SWE is presented. They correspond to rather unrealistic situations that have the purpose of illustrating the capabilities and/or the accuracy/inaccuracy achieved by the codes. The other benchmark solution is related to a real engineering problem that can be ambiguously modeled with the shallow water codes.

2 GOVERNING EQUATIONS

The depth-integrated Navier-Stokes equations of motion, previously averaged over turbulence, known as the SWE, are expressed in Cartesian form as[7]

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} + \frac{\partial \mathbf{G}}{\partial y} = \mathbf{H} \tag{1}$$

The vectors $\mathbf{U}$, $\mathbf{F}$, and $\mathbf{G}$ can be expressed in terms of the primary variables, u, v and h as

$$\mathbf{U} = \begin{bmatrix} h \\ uh \\ vh \end{bmatrix}, \quad \mathbf{F} = \begin{bmatrix} uh \\ u^2h + gh^2/2 - 2h\nu_t \frac{\partial u}{\partial x} \\ uvh - h\nu_t \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \end{bmatrix}, \quad \mathbf{G} = \begin{bmatrix} vh \\ uvh - h\nu_t \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \\ v^2h + gh^2/2 - 2h\nu_t \frac{\partial v}{\partial y} \end{bmatrix} \tag{2}$$

The source term $\mathbf{H}$ is given by

$$\mathbf{H} = \begin{bmatrix} 0 \\ -g\frac{\partial \eta}{\partial x} - \tau_{bx}/\rho \\ -g\frac{\partial \eta}{\partial y} - \tau_{by}/\rho \end{bmatrix} \tag{3}$$

In the above equations, (u, v) are the depth-averaged velocity components along the streamwise and lateral horizontal directions (x, y) respectively, t is the time, h is the total water depth, g is the acceleration of gravity, ρ is the water density, (τ_{bx}, τ_{by}) are the shear stresses components acting on the stream bed, η is the channel bed elevation, and ν_t is the turbulent eddy viscosity. For depth-averaged calculations, one often approximates the turbulent viscosity as $\nu_t \approx \alpha u_* h$, where u_* is the bottom friction velocity defined as $\sqrt{|\vec{\tau}_b|/\rho}$. The constant α, that may range from 0.07 to 0.30, not only takes into account the mixing process due to turbulence but also the vertical flow inhomogeneities. To close the problem, the classical squared function dependency on the depth-averaged velocity is used to model the bed resistance

$$\tau_{bx} = C_F \rho |u| u, \quad \tau_{by} = C_F \rho |u| v \tag{4}$$

Sometimes, the friction coefficient C_F is traditionally replaced by other coefficients such as the Manning or Chèzy relations

$$C_F = g n^2 h^{-1/3}, \quad C_F = g\, C^{-2} \tag{5}$$

where n is the Manning roughness coefficient, and C is the Chèzy discharge coefficient.[8]

3 SOME FINITE ELEMENT CODES BASED ON THE SWE

3.1 Codes description

In order to compare the analytical solutions with numerical results, the following codes have been tested:

- *RMA-2:* this code was developed by Norton, King and Orlob, of Water Resources Engineers, for the Walla Walla District, U.S. Army Corps of Engineers, and released in 1973. Further development, was carry out by King and Roig at the University of California, Davis. Subsequent enhancements have been made by King and Norton, of Resource Management Associates (RMA), and by the Waterways Experiment Station (WES) Hydraulic Laboratory.[4]

- *FESWMS-2DH:* the code was developed for the Federal Highway Administration by the Water Resources Division of the U.S. Geological Survey. FESWMS-2DH consists of three programs: a data-input module, DINMOD; a hydrodynamic flow module, FLOMOD; and an analysis-of-output module, ANOMOD. The preprocessor, DINMOD, generates a two-dimensional finite-element network for use by FLOMOD.[5] However, in this work both RMA-2 and FESWMS-2D are accessed through the user friendly interface SMS.[9]

- *TELEMAC-2D:* was developed by the National Hydraulics Laboratory (Laboratoire National d'Hydraulique - LNH) of the Research and Studies Directorate of the French Electricity Board (EDF-DER). The software is imbedded in an integrated and user-friendly software environment, the TELEMAC system.[6]

3.2 Solution techniques

Both RMA-2 and FESWMS-2DH use a fully implicit implementation of the Galerkin weighted residual technique to solve the governing system of differential equations.[5,10] In these codes, the values of a dependent variables are approximated by mixed interpolation, that is, the nodal values of water-depth are defined at three or four vertices and their distribution is approximated by linear or bilinear polynomials for triangular or rectangular elements, respectively, while the depth-averaged velocity are approximated at six or eight vertices, and so its distribution is interpolated with quadratic polynomials for triangular or rectangular elements, respectively. FESWMS-2DH includes, in addition, nine-node rectangular elements, as shown in Figure 1.

Owing to the non-linearity of the governing equations, the numerical integration of the Galerkin procedure is performed iteratively using an implicit Newton-Raphson procedure. Both RMA-2 and FESWMS fully assemble the matrices of the linear system at each iteration within an implicit scheme for the time integration process, which has the advantage of maintaining stability and mass conservation at Courant numbers in excess of 1. The resultant matrices are both large and sparse, and can add significantly to the time required to perform a particular calculation. Therefore, a suitable reordering scheme is necessary to reduce the matrix frontwidth.

TELEMAC-2D employs a fractional step method or splitting technique[11,12] to solve the governing equations (eqs. 1), where advection terms are initially solved with the Method of Characteristics (MOC),[13] and separated from diffusion and source terms, which are solved together in a second step. This is achieved for a space discretization consisting of linear triangles with three nodes (Figure 1).

The second step of TELEMAC-2D makes use of a time discretization of the predictor-corrector type and solves the resulting linear system with a conjugate gradient type method. In addition, the TELEMAC-2D code makes significant savings in both computational time and storage requirements through the use of an element-by-element solution technique. Here, the matrices of the linear system are stored in their elementary form

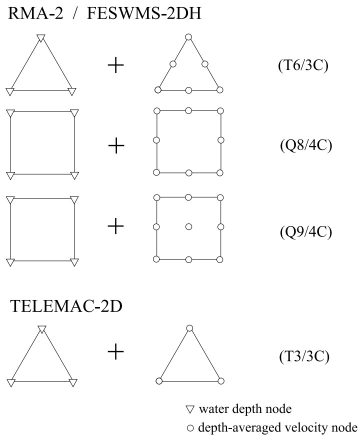

Figure 1: Depth-averaged velocity-water depth finite elements

without full assemblage. In summary, the numerical scheme followed by TELEMAC-2D is

- *1st step:* The advection terms are solved with the MOC, that is, if the spatial coordinate is 'convected' along the problem *characteristics*, the convective terms disappear and the remaining problem is that of simple diffusion for which standard discretization procedures are optimal[13]

$$\frac{h_* - h^n}{\Delta t} + \mathbf{u}^n \cdot \nabla h^n = 0 \quad (6)$$

$$\frac{\mathbf{u}_* - \mathbf{u}^n}{\Delta t} + \mathbf{u}^n \cdot \nabla \mathbf{u}^n = 0 \quad (7)$$

- *2nd step:* The remaining terms are now advanced on the basis of the 1st step solution h_*, $\mathbf{u}_*$. The mass conservation is, in consequence, discretized as follows

$$\frac{h^{n+1} - h_*}{\Delta t} + (h\,\nabla \cdot \mathbf{u}) = S^n \quad (8)$$

where S represents source/sink terms, if any, and the non-linear terms are linearized mathematically to remove the need for an iterative solution of the Newton-Raphson type. For example, the non-linear term above is linearized as

$$(h\,\nabla \cdot \mathbf{u}) = \widetilde{h}\,\nabla \cdot \left(\theta \mathbf{u}^{n+1} + (1-\theta)\mathbf{u}^n\right) \qquad 0 \le \theta \le 1 \quad (9)$$

where θ is an implicit weighting coefficient bounded between zero and one, and the superscripts n and $n+1$ indicate the time step level,[12] and

$$\widetilde{h} = \begin{cases} h^n & \text{without sub-iteration loop} \\ h^{n+1} & \text{with a sub-iteration loop} \end{cases} \tag{10}$$

The momentum conservation equation is then advanced as follows:

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}_*}{\Delta t} = -g\nabla H^{n+1} + \mathbf{F}^n + \nu_t \nabla^2 \mathbf{u}^n \tag{11}$$

where

$$-g\nabla H^{n+1} = -g\nabla(h^{n+1} - h^n) - g\nabla H^n \tag{12}$$

The resultant system of linear equations are solved with the GMRES solution algorithm (Generalised Minimum RESidual). Further details about the GMRES algorithm implementation can be consulted in Reddy and Gartling.[14]

Finally, the governing equations are formulated in a weak sense using the Method of Weighted Residual (MWR), where the water-depth and the velocity fields are approximated using the standard FEM basis functions

$$h \approx \sum_{j=1}^{N} h_j(t)\phi_j(\mathbf{x}), \qquad \mathbf{u} \approx \sum_{j=1}^{N} \mathbf{u}_j(t)\phi_j(\mathbf{x}), \tag{13}$$

where N is the number of discrete points, and $h_j(t)$ and $\mathbf{u}_j(t)$ are the nodal unknowns ($j = 1, \ldots, N$) of the water-depth and the depth-averaged velocity components, respectively, and $\phi_j(\mathbf{x})$ are the approximation functions defined over the finite elements described in Figure 1. Then, the solution is advanced in time as just explained.

4 BENCHMARKS SOLUTIONS

In this short communication, uncertainty in model parameterization and boundary conditions, and complex topography are ignored.

4.1 Slowly divergent channel

Restricting the analysis to the 1D case, and zero viscosity ($\nu_t = 0$), the equations (1) are reduced to

$$\begin{aligned} h_t + (uh)_x &= 0 \\ (hu)_t + (hu^2 + g\frac{h^2}{2})_x &= gh(S_0 - S_f), \end{aligned} \tag{14}$$

where S_0 is the bed slope and S_f is the friction slope. It is then possible to find a solution to this system of equations if it is assumed that the flow takes place in a slowly divergent channel, such that two-dimensionality can be safely ignored, that satisfies

$$b(x)/b_0 = \frac{1+\epsilon}{\sqrt{1+[(1+\epsilon)^2-1]\mathrm{e}^{-2C_F\,x/h_0}}} \tag{15}$$

where $b(x)$ is the channel width, measured from b_0 at the entrance $x = 0$, and ϵ is an arbitrary coefficient that measures the rate of geometric divergence of the channel. Thus, if now the flow experiences a sudden jump in its inflowing momentum at $x = 0$, the solution can be expressed in terms of the base flow u_0 and h_0 as

$$u(x)/u_0 = \sqrt{1+\left[\left(\frac{u_A^2}{u_0^2}\right)-1\right]\mathrm{e}^{-2C_F\,x/h_0}}, \tag{16}$$

where

$$u_0 = \left(\frac{g\,h_0\,S_0}{C_F}\right)^{1/2}. \tag{17}$$

Above, u_A when subtracted from u_0 represents the given excess of momentum. The values used in this test were $C_F = 0.003924$, $h_0 = 10$ m, $u_0 = 1$ m/s, $u_A = 1.5$ m, $\epsilon = 0.5$, and $b_0 = 600$ m.
The finite element meshes used for the calculations comprises 707 nodes and 1200 elements for TELEMAC-2D, and 2013 nodes and 600 elements for both RMA-2 and FESWMS-2DH. The computational mesh is shown in Figure 2. Figure 3 plots the comparison between the analytical and numerical solution of u/u_0 obtained with the different codes. The maximum relative error of h and u are shown in Table 1. As it can be seen, all codes results are in good agreement when compared with the analytical solution, although FESWMS-2DH exhibits the major discrepancy at the channel inflow.

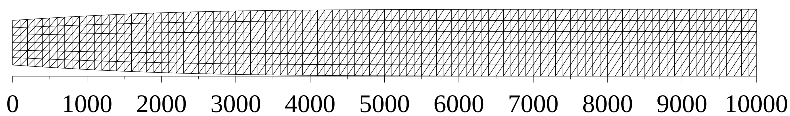

Figure 2: Finite element mesh used for the slowly divergent channel test

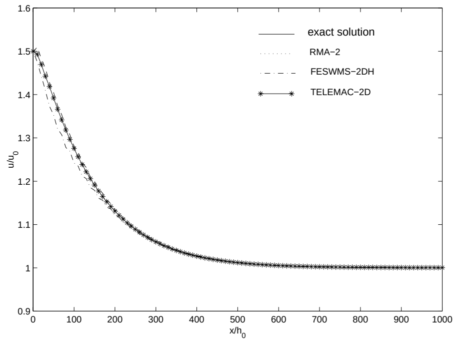

Figure 3: Comparison between analytical and numerical solution of u

Numerical code	% error (h)	% error (u)
RMA-2	0.187	3.61
FESWMS-2DH	0.120	11.27
TELEMAC-2D	0.108	2.51

Table 1: Maximum relative error in h and u, computed as $\max |f_{num} - f_{exac}|/f_{exac}$

4.2 Flow in an annular channel

The second problem considered here is the fully developed, uniform flow in a circular (annular) channel of rectangular cross-section. The down reach slope $S_0 = -d\eta/d\theta$ is constant, where $\eta = \eta(\theta)$ represents the channel bottom elevation in terms of the angular coordinate θ. Then, for simple geometrical reason, the actual slope S of the channel satisfies the equation $S = S_0/r$. Under these circumstances, all derivatives with respect to θ dropped from now on, and the equations of motions in polar coordinates (r, θ) reduces to

$$\frac{d(hu_r)}{dr} = 0 \tag{18}$$

$$-\frac{u_\theta^2}{r} + g\frac{dh}{dr} = 0 \tag{19}$$

$$\frac{g}{r}S_0 - C_F\frac{u_\theta^2}{h} = 0, \tag{20}$$

where $h = h(r)$ is the water depth, and $u_r = u_r(r)$, $u_\theta = u_\theta(r)$ are the radial and tangential velocity components, respectively.
The equations (19-20) can be interpreted physically: the first equates the free surface slope

in the radial direction with the centripetal acceleration required to maintain the circular motion of the fluid, the second balances gravity against bed friction. The analytical solutions can be expressed in terms of the Froude number $F_0 = \sqrt{S_0/C_F\, r_0}$, and the dimensionless radii $\widetilde{r} = r/r_0$, relative to the inner radii r_0

$$\frac{u_\theta}{u_\theta^0} = \mathrm{e}^{-F_0^2(1-\widetilde{r})/2\widetilde{r}}/\sqrt{\widetilde{r}} \tag{21}$$

$$\frac{h}{h_0} = \mathrm{e}^{-F_0^2(1-\widetilde{r})/\widetilde{r}} \tag{22}$$

where $u_\theta^0 = u_\theta(r_0)$, $h_0 = h(r_0)$ are the tangential velocity and the water depth at the inner radii $\widetilde{r} = 1$, respectively. For this test the following values were used: $r_0 = 50$ m, $h_0 = 1$ m, and Chèzy friction coefficient $C = 40$ m$^{1/2}$s^{-1}, related with C_F by g/C^2. The channel was 20 m wide. The boundary conditions imposed were the u_θ velocity component and the water depth h at the upstream end (flow was in counterclockwise direction), and the water depth h at the downstream end. These values were prescribed with the aid of (21) and (22). Figure 4 plots some of the meshes used in the simulations. Details about the spatial discretization used by the computational codes are detailed in Table 2. In this test, meshes A and B are structured and mesh C is non-structured.

Figure 5 shows the comparison of the numerical results for the depth and the u_θ-component of the velocity. The RMA-2 and TELEMAC-2D models reproduce accurately the depth distribution while FESWMS-2DH shows the major departure. However, some discrepancies are observed in the computation of the flow velocity. In this case, the FESWMS-2DH velocity solution does not converge. The effect of the mesh refinement is shown in Figure 6, where only the TELEMAC-2D solutions are included.

	mesh A		**mesh B**		**mesh C**	
	1	2	1	2	1	2
number of nodes	585	165	2193	585	3233	877
number of elements	256	256	1024	1024	1480	1480

Table 2: Finite element mesh data, 1: RMA-2 and FESWMS-2DH; 2: TELEMAC-2D

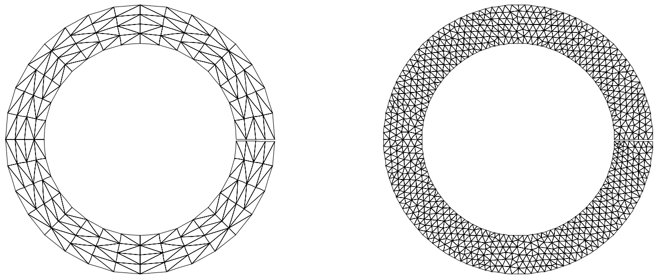

Figure 4: Left: Structured mesh of annular channel. Right: Non-structured mesh of annular channel

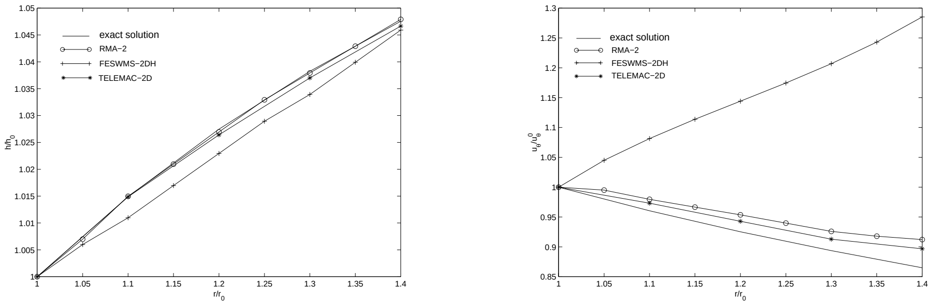

Figure 5: Left: Comparison between analytical and numerical solution of $h/h0$. Right: Comparison between analytical and numerical solution of u_θ/u_θ^0

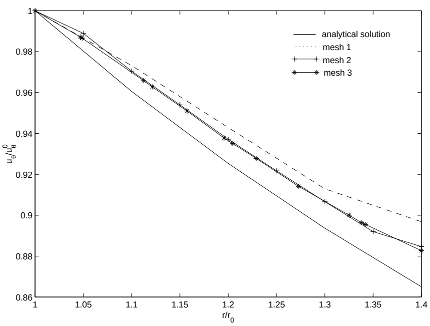

Figure 6: Effect of mesh resolution on the numerical computations

4.3 Branching channel

This test is designed to illustrate how a real engineering problem may be ambiguously model with some shallow water codes. It is related with the branching problem that frequently occurs in the main channel of a large river, such as the Paraná river in Argentina. The main conclusions of this test will be further discussed at the time of the Conference. In the meantime, an intensive field data collection work has been undertaken by the research group at FICH using an ADCP (Accoustic Doppler Current Profiler) for accurate measurements of the river discharge at selected locations of the Paraná river nearby Santa Fe city. One of these data set collected by the group in the middle reach of the Paraná river is discussed next.

5 COMPARISON OF NUMERICAL RESULTS WITH FIELD DATA

Nearby the Santa Fe and Paraná cities, Argentina, the middle reach of the Paraná river constitutes a large alluvial stream that flows through a broad plain (Figure 7). The river comprises a sand bed main channel, approximately 1000 m wide and 10 m deep, bounded by a steep, stable bank on the Paraná' side, and by a 10-15 km wide floodplain on the Santa Fe' side, with a mean discharge of the order of 17000 m^3/s.

In order to validate the numerical codes, the flow rate was measured in three cross sections placed in a bifurcation around two islands where the river splits in three flow branches (Figure 7). The finite element mesh spans about 10 km consisting of 4330 quadratic triangular elements with 9045 nodes (Figure 8). The mesh was refined in the area surrounding the two islands in order to capture the steep slope of the left bank. The condition of steady state flow at bankfull discharge was assumed for the simulations. At the inflow boundary was imposed a know discharge, and at the outflow a fixed water surface elevation. In addition, the slip boundary condition was automatically satisfied on all the rigid boundaries of the computational domain. Some preliminary results obtained with the RMA-2 and TELEMAC-2D are summarized in Table 3. It can be seen that the major discrepancy is obtained in Branch 2. However, this difference can be mainly attributed to the lack of a proper representation of the bed topography in this river section (at the time of the field work, the river was in low-water condition making it impossible to gauge the bed with an echosounder). The same calculation was done with the TELEMAC-2D (Table 3). Surprisingly, an error of the order of 30 % is obtained in Branch 3.

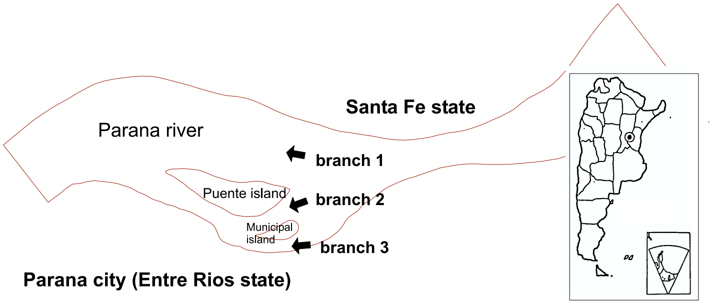

Figure 7: Location of the middle reach of the Paraná river in Argentina

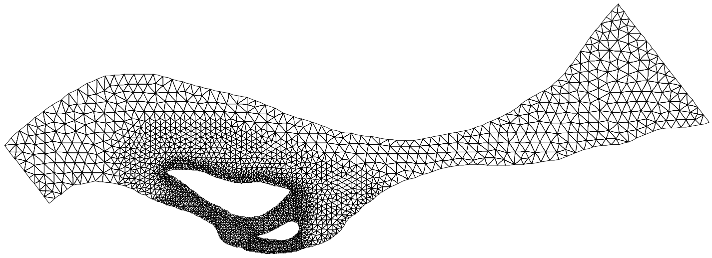

Figure 8: Finite element mesh

		RMA-2		TELEMAC-2D	
	Q_{obs}	Q_{sim}	% error	Q_{sim}	% error
Branch 1	9900.	9924.	0.24	10486.	5.92
Branch 2	213.	305.	43.19	248.	16.43
Branch 3	1729.	1639.	5.20	1157.	33.08

Table 3: Comparison of the bulk discharge in a branching point of the main channel of the Paraná river

6 CONCLUSIONS

Some computational codes that calculate free surface flows in water bodies have been tested against benchmark solutions. The codes tested showed good results in the case of the slowly divergent straight channel. Though not shown here, the RMA-2 and FESWMS-2DH solutions for the water-depth showed the typical checkerboard pattern of the mixed

FEM formulation.[15] For the case of the annular channel, both RMA-2 and TELEMAC-2D results showed a reasonable agreement with the exact solution of $h = h(\theta)$, but with a discrepancy of the order of 5 % for the outer value of the u_θ component of the flow velocity. In this test, the FESWMS-2DH solution did not converge for the finer mesh, while exhibited convergence for the h-space solution only for the coarse mesh. For the field study case, the TELEMAC-2D shows the greater discrepancy in determining the streamflow discharge at the river bifurcation. Therefore, in spite of their simplicity, the benchmark solutions proposed in this work have the merit to highlight some deficiencies of popular numerical engines for solving the SWE. Four more benchmark solutions are underway for a more extensive testing of the aforementioned codes. It is the purpose of the authors to have some of them ready at the time of the Conference. One of it is being specially designed to explain the lack of accuracy of the streamflow distribution at river bifurcations computed by TELEMAC-2D. This issue is of particular relevance in a movable bed configuration, i.e., when bed erosion and sediment transport is allowed in the modeling process.

7 ACKNOWLEDGEMENTS

This work has been supported by the Consejo Nacional de Investigaciones Científicas y Técnicas, Argentina (CONICET) and by the Universidad Nacional del Litoral, Argentina.

REFERENCES

[1] Vreudenhill C.B. *Numerical Methods for shallow-water flows.* Publishers, The Netherlands, (1998).

[2] Bates P.D. and Anderson M.G. A two-dimensional finite-element model for river flow inundation. *Proc. R. Soc. Lond. A*, **440**, 481–491 (1993).

[3] Lane S.N. Hydraulic modelling in hydrology and geomorphology: A review of high resolution approaches. *Hydrological Processes*, **12**, 1131–1150 (1998).

[4] *User guide to RMA2-WES Version 4.3.* U.S. Army Corps of Engineers – Waterways Experiment Station Hydraulics Laboratory, (1997).

[5] Lee J.K., Froehlich D.C., Gilbert J.J., and Wiche G.J. A two–dimensional finite element model study of backwater and flow distribution at the I–10 crossing of the Pearl River near Slidell, Lousiana. USGS Water Res. Inv. Report 82-4119, 66 p, (1993).

[6] *2D Hydrodynamics TELEMAC-2D software, v.4.0, User's Manual.* EDF. Electricité de France, Département Laboratoire National d'Hydraulique, (1998).

[7] Weiyan T. *Shallow water hydrodynamics - Mathematical Theory and Numerical Solution for a two-dimensional system of shallow water equations.* Elsevier Oceanography Series, 55, (1992).

[8] Chow V.T. *Open Channel Flow.* McGraw–Hill: New York, (1959).

[9] *Surfacewater Modelling System User's Manual.* Brigham Young University, Utha,

USA., (1999).

[10] King I.P. and Norton W.R. Recent applications of RMA's finite element models for two dimensional hydrodynamics and water quality. In C.A. Brebbia, W.G. Gray, and G.F. Pinder, editors, *Proceedings of the Second International Conference on Finite Elements in Water Resources*, pages 81–99, Pentech Press, (1978).

[11] Marchuk G.I. *Methods of Numerical Mathematics,* 2nd ed. Springer-Verlag, (1982).

[12] Hervouet J.M-. and Van Haren L. Recent advances in numerical methods for fluid flows. In M G Anderson, D E Walling, and P D Bates, editors, *Floodplain Processes*, pages 183–214, John Wiley and Sons: Chichester, (1996).

[13] Zienkiewicz O.C. and Taylor R.L. *The finite element method*, volume III. McGraw Hill, (2000).

[14] Reddy J.N. and Gartling D.K. *The finite element method in heat transfer and fluid dynamics,* 2nd ed. CRC Press LLC, (2001).

[15] Brezzi F. and Bathe K.J. A discourse on the stability conditions for mixed finite element formulations. *Comput. Methods Appl. Mech. Engrg.*, **82**, 27–57 (1990).