Numerical Simulation of Tidal Currents in Persian Gulf

Ameleh Aghajanloo, Moharam Dolatshahi Pirouz, Masoud Montazeri Namin

Abstract—In this paper, a two-dimensional (2D) numerical model for the tidal currents simulation in Persian Gulf is presented. The model is based on the depth averaged equations of shallow water which consider hydrostatic pressure distribution. The continuity equation and two momentum equations including the effects of bed friction, the Coriolis effects and wind stress have been solved. To integrate the 2D equations, the Alternative Direction Implicit (ADI) technique has been used. The base of equations discritization was finite volume method applied on rectangular mesh. To evaluate the model validation, a dam break case study including analytical solution is selected and the comparison is done. After that, the capability of the model in simulation of tidal current in a real field is represented by modeling the current behavior in Persian Gulf. The tidal fluctuations in Hormuz Strait have caused the tidal currents in the area of study. Therefore, the water surface oscillations data at Hengam Island on Hormoz Strait are used as the model input data. The check point of the model is measured water surface elevations at Assaluye port. The comparison between the results and the acceptable agreement of them showed the model ability for modeling marine hydrodynamic.

Keywords—Persian Gulf, Tidal Currents, Shallow Water Equations, Finite Volumes

I. INTRODUCTION

DUE to increasing issues related to marine currents such as marine environment, utilization of marine energy, and also the transmission of pollution in seas and oceans, today coastal and marine engineers consider simulation of currents hydrodynamic behavior in such environments. Getting to know the current status and water surface oscillation in these regions precisely may be a powerful tool to face with different marine phenomena. Moreover, because of rapidly growing of computer technology and numerical techniques in fluid dynamic simulation, the computational fluid dynamics (CFD) has been suitable and applicable tool for modeling the complex hydraulic and marine engineering problems.

Since there are significant problems in three dimensional (3D) hydrodynamic modeling, it is preferred to use the methods through which it is possible to save computational time and cost to an acceptable extent without entering too many errors in results. The two-dimensional (2D) shallow water equations (SWE) describe the free surface flows in open channels. Also, it is suitable for the cases in which the vertical acceleration is assumed to be small compared to the gravitational acceleration; the hydrostatic pressure distribution

K. Aghajanloo is with the University of Tehran, Tehran, Iran. (phone: +989125871157; fax: +98216111; e-mail: aghajanloo@ut.ac.ir).

M. D. Pirooz is with University of Tehran, Tehran, Iran. (e-mail: mdolat@ut.ac.ir).

M. M. Namin is with University of Tehran, Tehran, Iran. (e-mail: mnamin@ut.ac.ir).

assumption is made for the governing equations of fluid flow. The system of SWE is a time-dependent set of non-linear partial differential equations of hyperbolic type.

Many researchers have conducted studies in solving shallow water equations whose most difference is in techniques applied in equations discretization, applied mesh generator and also the terms applied in equations set.

Depth-integrated 2D hydrodynamic models based on a regular grid have been used for many years for predicting free surface flows and several techniques are available in literature to solve the problems. General approaches to discreitize the equations include finite difference methods (FDM) [1]-[3], finite element methods (FEM) [4] and the finite volume methods (FVM) [3], [5]. FVM is the most common method for solving problems related to bioenvironmental flows and is applied in marine and ocean environments, in open channels [5] or in more accurate problems such as dam break [6]. Indeed, FVM is an integration of simplicity of FDM and geometric flexibility of FEM which caused this method to be applied for all structured and unstructured mesh. According to studies done as well as the significant advantages of FVM, the method was used as discreitization equations base in this study [17], [18].

The Persian Gulf is an aquatic environment always considered as a strategic region both for marine transportation and transferring fuels obtained from great oil fields to other parts of the world. These issues caused different marine pollution such as great oil spill to happen during the current years. In this paper, a mathematical model has been provided to simulation the hydrodynamic behavior of the tidal currents. The governing equations have been discritized on a staggered rectangular mesh and based on the finite volume method for spatial discritization and the forward finite difference for temporal discritization.

To solve 2D equations in x and y directions, the Alternative Direction Implicit (ADI) method is used. The ADI method was introduced by Peaceman and Rachford [9] and Douglas and Rachford [10] and then was generalized by Douglas and Gunn [11]. Due to its large stability domain and significant balance between computational cost and accuracy, this method has been widely used to solve SWEs [12]. In this technique, the direction of solving equations is exchanged between x and y directions alternatively and an integration of momentum and continuity equations are solved in each one dimensional sweeps which may reduce the complexity of equations to a reasonable extent. The application of the ADI scheme results in a tridiagonal system of equations for each grid line. The solution is obtained by inverting the tridiagonal matrix using the Double Sweep (DS) algorithms, a very fast and accurate

form of Gauss elimination.

This model is verified for several test cases that one of them including currents due to dam break in a straight channel is represented here and the capability of model is evaluated.

In the more progressive stages of research, the results of this 2D hydrodynamic model are used in an environmental model (oil spill model) as presented in next papers of authors.

II. GOVERNING EQUATIONS

The mathematical method which was used for the study is a two-dimensional shallow water equation (SWE). We reach the model by computing the average of the Navier-Stokes equations along vertical direction with regard to some assumptions. The most important factors are the smallness of the vertical velocity component compared with the horizontal ones and the hydrostatic distribution of pressure. What follow below is the equation and their representation:

$$\frac{\partial \xi}{\partial t} + \frac{\partial p}{\partial x} + \frac{\partial q}{\partial y} = \frac{\partial z}{\partial t} \qquad (1)$$

$$\frac{\partial p}{\partial t} + \frac{\partial}{\partial x}\left(\frac{p^2}{h}\right) + \frac{\partial}{\partial y}\left(\frac{pq}{h}\right)$$
$$= -gh\frac{\partial \xi}{\partial x} + \frac{\partial}{\partial x}\left(\upsilon_t \frac{\partial p}{\partial x}\right) + \frac{\partial}{\partial y}\left(\upsilon_t \frac{\partial p}{\partial y}\right) \qquad (2)$$
$$- \frac{\tau_{bx}}{\rho_w} + \frac{\tau_{sx}}{\rho_w} - f_{cx}$$

$$\frac{\partial q}{\partial t} + \frac{\partial}{\partial x}\left(\frac{pq}{h}\right) + \frac{\partial}{\partial y}\left(\frac{q^2}{h}\right)$$
$$= -gh\frac{\partial \xi}{\partial y} + \frac{\partial}{\partial x}\left(\upsilon_t \frac{\partial q}{\partial x}\right) + \frac{\partial}{\partial y}\left(\upsilon_t \frac{\partial q}{\partial y}\right) \qquad (3)$$
$$- \frac{\tau_{bx}}{\rho_w} + \frac{\tau_{sx}}{\rho_w} + f_{cy}$$

where h is flow depth; p and q are velocity fluxes in x and y directions ($p = h \cdot u$ and $q = h \cdot v$); u and v are depth averaged flow velocities in x and y directions; ξ is water surface elevation ($\xi = h + z$); g is gravitational acceleration, υ_t is eddy viscosity coefficient, f_{cx} and f_{cy} are Coriolis forces due to earth rotation $f_{cx} = -q\Omega$ and $f_{cy} = p\Omega$. τ_{bx}/ρ_w and τ_{by}/ρ_w are bed friction stresses term in x and y calculated using ($\tau_{bx}/\rho_w = gp\sqrt{p^2+q^2}/C^2h^2$ and $\tau_{bq}/\rho_w = gq\sqrt{p^2+q^2}/C^2h^2$), C is Chezzy Coefficient; τ_{sx} and τ_{sy} are wind stresses in x and y calculated using ($\tau_{bx} = \rho_{air}C_D u_w |U_{wind}|$ and $\tau_{by} = \rho_{air}C_D v_w |U_{wind}|$), ρ_{air} and ρ_w are air and water density, respectively, C_D is wind drag coefficient, U_{wind} is wind speed on 10m over water surface, u_w and v_w are wind speed components in x and y directions ($U_{wind} = \sqrt{u_w^2 + v_w^2}$).

In this model, the eddy viscosity coefficient could be computed by two types of turbulent models. The constant coefficient formulation that is in widespread use in hydrodynamic models which is presented as $\upsilon_{th} = \alpha_t U_* h$ where the bed friction velocity is defined as $U_* = \sqrt{C_f(U^2 + V^2)}$ and the empirical coefficient α_t is directed to be between 0.3 and 1.0 [14]. Another applicable model is SGS [15] in that the eddy viscosity coefficient is calculated as $\upsilon_{th} = C_s^2 \Delta^2 |\overline{S}|$ where C_s is constant and taken to be 0.1 to 1.0; Δ^2 presents the grid size which can be consigned by the area of the calculated cell. $|\overline{S}| = \sqrt{\overline{S}_{ij}\overline{S}_{ji}}$ and $\overline{S}_{ij}$ is the tensor form and could be calculated by the following equation [16]:

$$\overline{S}_{ij} = \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \qquad (4)$$

These turbulence models are known suitable for depth averaged equations and have been used in some similar applications.

III. NUMERICAL FORMULATIONS

A. Mesh Generator

In order to accurately implement this method for real problems where we face many difficulties in coastline and also great different variation and modification in bed elevation, a sub program which can define such complexities for the model is included in hydrodynamic model.

At the beginning, as input data, the data and places of a set of lines and curves are given to the model which in fact represent coast and island borders, Mesh generator model determines the boundaries of the domain by using this information. The next stage will determine the Bathymetry, and then the bed elevation is calculated for each grid by interpolation which uses the bed elevation contour lines of the domain. Therefore, the model is capable of simulating all domains from experimental case studies with simple geometry to real domain with complex geometry.

B. Solution Techniques of Governing Equations

In order to solve the governing equations numerically, finite volume method (FVM) has been applied for spatial discretization and forward finite difference has been utilized for temporal discretization. The different terms are represented on a staggered grid in x-y space which is shown in Figure 1.
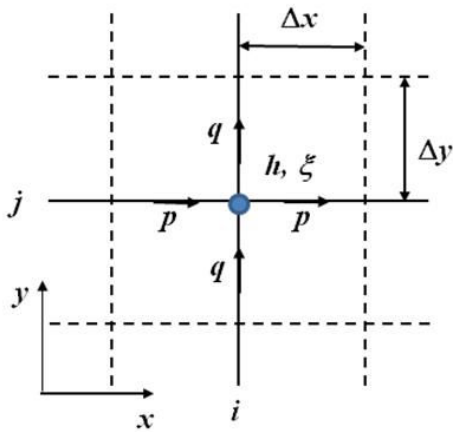
Fig. 1 Difference Grid in x-y space

Since there are variations of existing terms in the governing equations and the deployed method to solve them, it is preferred to use a method which provides the solution with the possibility of choosing a different way to solve each term according to their nature. This method will finally reach to an acceptable and appropriate solution for equations set. Time splitting technique may be suitable for this purpose. By this technique, and with regard to the number of available terms and the domain dimensions, each time step is divided into some virtual time steps. Therefore, through a separate method, all terms will be solved and could be different from others. Hence, it gives the ability to solve different terms by a suitable and perfect method [17].

To solve the transport equation, including advection and diffusion terms in finite volume methods, the passed fluxed from computation cell boundaries (upstream and downstream) should be computed. The change of parameter ψ (in each equations, ψ which has different description) in each time step is the difference of these fluxes.

$$\frac{\partial \psi}{\partial t} + \frac{\partial (u\psi)}{\partial x} = \frac{\partial}{\partial x}\left(D\frac{\partial \psi}{\partial x}\right) \qquad (6)$$

$$\psi_{i,j}^{n+1} = \psi_{i,j}^{n} - \frac{\Delta t}{\Delta x}\left(F_{i+\frac{1}{2},j}^{total} - F_{i-\frac{1}{2},j}^{total}\right) \qquad (7)$$

where F^{total} is a combination of fluxes due to the advection and diffusion mechanisms ($F^{total} = F^{Advection} + F^{Diffusion}$). The Fromm scheme, two-order accuracy method, has been applied to solve the advection terms [18]. It has been constructed by assuming a linear distribution of concentration in the control volumes that defines the flux as:

$$\Delta t(u.\psi)_{i+1/2,j} = u\Delta t\left[\psi_{i,j} + \frac{1}{2}(\Delta x - u\Delta t)S_{i,j}\right] \qquad (8)$$

In that $S_{i,j} = (\psi_{i+1,j} - \psi_{i-1,j})/2\Delta x$ for $u > 0$.

To compute the diffusion term, the semi-implicit approach is used. By this method that can be expressed by:

$$\begin{aligned} \psi_{i,j}^{n+1} = \psi_{i,j}^{n} \\ + \frac{\Delta t \cdot D_x}{2(\Delta x)^2}\left(\psi_{i+1,j}^{n+1} - \psi C_{i,j}^{n+1} + \psi_{i-1,j}^{n+1}\right) \\ + \frac{\Delta t \cdot D_x}{2(\Delta x)^2}\left(\psi_{i+1,j}^{n} - 2\psi_{i,j}^{n} + \psi_{i-1,j}^{n}\right) \end{aligned} \qquad (9)$$

The use of this equation in a one dimensional sweep, for a sequence of grid points leads to a three diagonal matrix:

$$A_i^1\psi_{i-1}^{n+1} + A_i^2\psi_i^{n+1} + A_i^3\psi_{i+1}^{n+1} = A_i^0 \qquad (10)$$

Then, it is solved by the well-known DS algorithm. Also, this method is used to compute the water elevations. The straight forward approximation has been chosen to obtain the other equations terms including gravity and bed friction terms.

In any numerical simulation, it is necessary to first make sure about the results accuracy of the model and then use it to solve the real problems. Therefore, a model for several simpler test samples with analytical solutions or laboratory measurements will be implemented and the results will be compared. For better results evaluation, error measurement criteria such as Relative Mean Absolute Error (RMAE) [19]-[21] and Root-Mean Square error (RMS) [22] will be utilized which are defined as follows:

$$RMAE = \frac{\sum_{i=1}^{N}\left|\Phi_{m(i)} - \Phi_{c(i)}\right|}{\sum_{i=1}^{N}\Phi_{m(i)}} \qquad (11)$$

$$RMS = 100\frac{\sqrt{\sum\left(\Phi_{m(i)} - \Phi_{c(i)}\right)^2}}{\sum\Phi_{m(i)}} \qquad (12)$$

The qualification for RMAE ranged is presented in Table 1 [19]:

TABLE I
QUALIFICATION OF ERROR RANGES FOR RMAE

Time	RMAE
Excellent	<0.2
Good	0.2-0.4
Reasonable	0.4-0.7
Poor	0.7-1.0
Bad	>1.0

IV. BOUNDARY CONDITIONS

In the solution process, the boundary conditions must introduce flux passed from adjacent boundaries of network points. General types of boundary conditions are applied in this model; the input, output and wall boundary conditions.

Input boundary conditions could be applied by imposing the water surface level fluctuations at tidal flow boundary and by importing a certain discharge to the domain. At the open boundaries, information such as surface levels or flux densities perpendicular to the boundary, to present the force outside the domain, must be allowed to be propagated into the domain.

A free-slip velocity boundary condition at walls can be imposed where no flow passes through a vertical plane of the flow domain. This is the condition for straight borders of the side-wall expansion test case. At these boundaries, the normal component of the fluid velocity is set to zero; therefore, the tangential velocities are maintained by using free-slip conditions at wall boundaries. No-slip boundary conditions could be imposed at rough wall boundaries. At these boundaries all velocity components are set to zero.

V. MODEL VERIFICATION

In this research, to demonstrate the validation of the model, a idealized dam-break test case with known analytical results was selected. A 2D channel with 6000m length and 600m width has been assumed to be frictionless, having a dam in the middle and with water depths to the upstream and downstream of the dam, at 3.2 m and 0.32 m, respectively. (see Fig. 2).
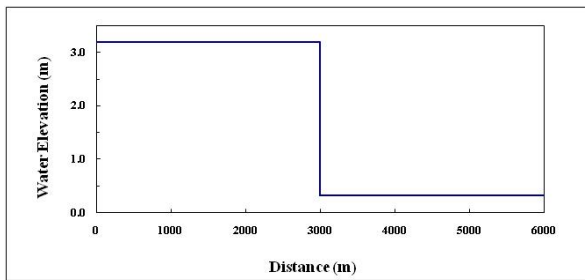

Fig. 2 Schematic presentation of initial condition of dam-break test case

The Channel is covered with a rectangular grid with cell dimensions of 50m. Thus, a network with 7503 points and 7320 cells has been established. Next, the input data and the network generated were defined for the mathematical model and the model time step of $\Delta t = 5\sec$ for a simulation time equal to 320 seconds was conducted. Now, the next step was to compare the results for which the deep water in the Flume midline was used. In Figure 3, the three dimensional view of water elevation for three modes of initial condition (before dam break), 120 sec after dam break and 300sec after dam break is shown.

As shown, the computed results of hydrodynamic model conform to analytical solution. The error criteria including RMAE and RMS at different times of 60, 120, 180, 240 and 300 seconds are demonstrated in Table 2. Based on the obtained results, All of RMAE criteria are less than 0.2 and all of RMS criteria are less than 2%. Therefore, the model can be used to simulate the more complex and real world problems.
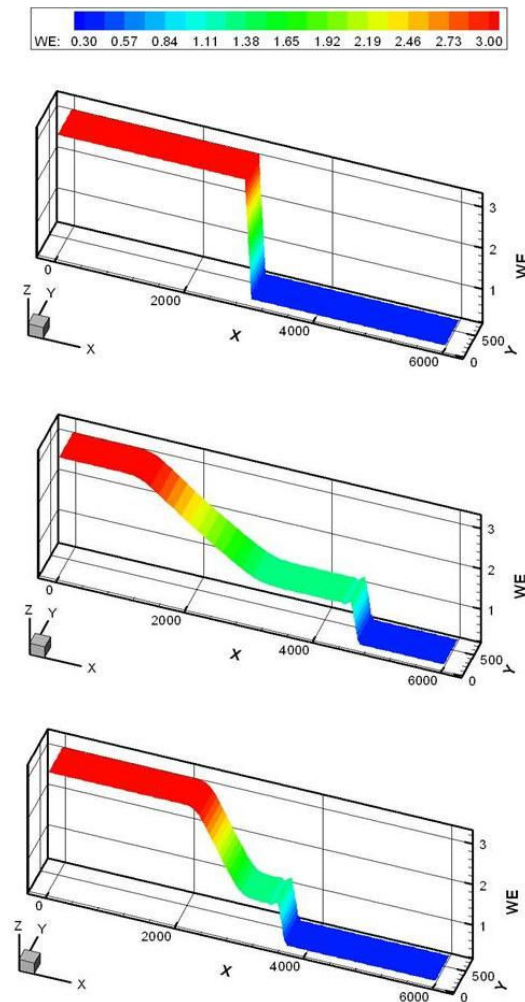

Fig. 3 3D view of water elevation (a) initial condition (before dam break) (b) 120 sec after dam break (c) 300 sec after dam break.

TABLE II
COMPUTED ERROR CRITERIA

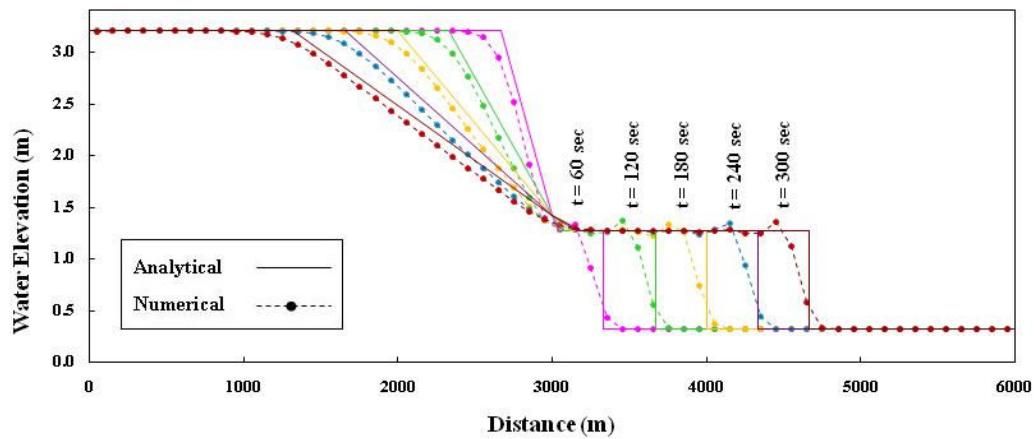| Time	RMAE	RMS (%)
60 *sec*	0.016	0.628
120 *sec*	0.022	0.822
180 *sec*	0.021	0.662
240 *sec*	0.022	0.546
300 *sec*	0.085	1.991

Fig. 4 The comparison between the model results and analytical solution in 60 sec time interval

VI. REAL WORD APPLICATIONS

The hydrodynamic model has been used to simulate the tidal currents in the Persian Gulf after evaluation of the ability of the model.

The Hormuz Strait is the major inflow boundary of The Persian Gulf which is located on the southern part of Iran and it is a closed marine environment (Fig. 5). It has about 1000km length, and 340 km width. The average depth of the Gulf is 36m. It covers a surface area of ~239,000km^2. Extensive shallow regions, <20m deep, are located along the coastlines of the United Arab of Emirates (hereafter referred to as the Southern Shallows), around Bahrain, and at the head of the Gulf. Deeper portions, >40m deep, are found along the Iranian coast stretching into the Strait of Hormuz, with a width of 56km that connects the Persian Gulf via the Gulf of Oman to the northern Indian Ocean. The only linking way of the Gulf to Open Seas is through this Strait; therefore, ocean conditions are imposed to this region through this border.

A. Geometry Modeling

In order to accurately simulate the hydrodynamic behavior flow, the geometrical characteristics of flow domain must be modeled precisely. In the first phase, the domain plane geometry is introduced to be modeled by several boundary curves and the model covers the area by a structured rectangular mesh including 58417 nodes and 57401 elements (Figure 6). But this is only the horizontal geometry. The Persian Gulf has also different bed elevation. For 3D modeling of the study area, we should include the topography, too. To do so, the 2D available data are used and the bed elevation for each node has been computed by interpolation of the surrounding nodes (Figure 7). In Figure 6, the location of Hormuz Strait is shown.

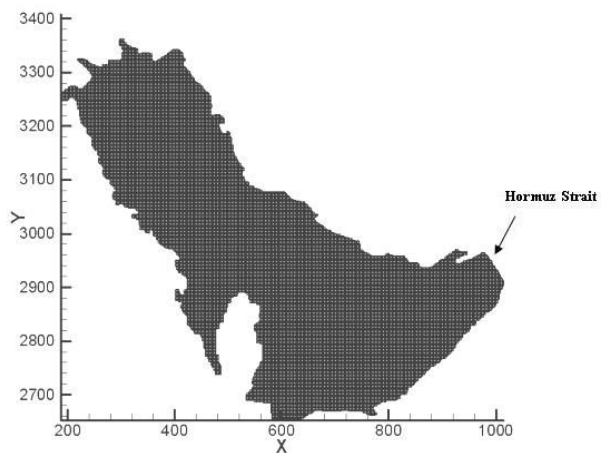
Fig. 5 The modeling study area, Persian Gulf

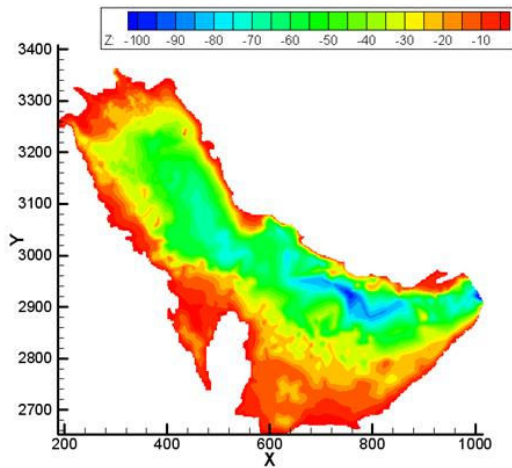Fig. 6 The computational mesh of study area

Fig. 7 The Bathymetry of study area

VII. IMPLEMENTATION OF HYDRODYNAMIC MODEL

Input boundary of the study area and the required data for water surface fluctuations in certain period of time are available and plotted in Figure 8 from 2002/10/1 to 2002/10/12 for imposing the tidal flow boundary condition at Hormuz Strait. After providing the model input data, it could be applied to simulate the tidal current in Persian Gulf. This test period covers 12 days and the tidal measurements are undertaken during fall tides with a maximum water level equal to +1.3m and minimum water level equal to -1.7m. To evaluate the model output, a check point has been chosen to compare the results of computed water level oscillation with observed data. This point is Assaluye Port, in the northern coastline of the Gulf. The comparison of results and measured data are depicted in Figure 10. When comparing the obtained results with the observed data, a good agreement between the two is observed. As it is shown in the figure, at the beginning days of simulation, some incongruence are observed in the results but it represents proper prediction of water surface level oscillations because of the warming up of the time and stability of model simulation process which shows a good capability of numerical model in flow simulation in real marine environments with complex geometry. Figure 9 shows the Computed results of water surface elevation several hours after simulation start time.
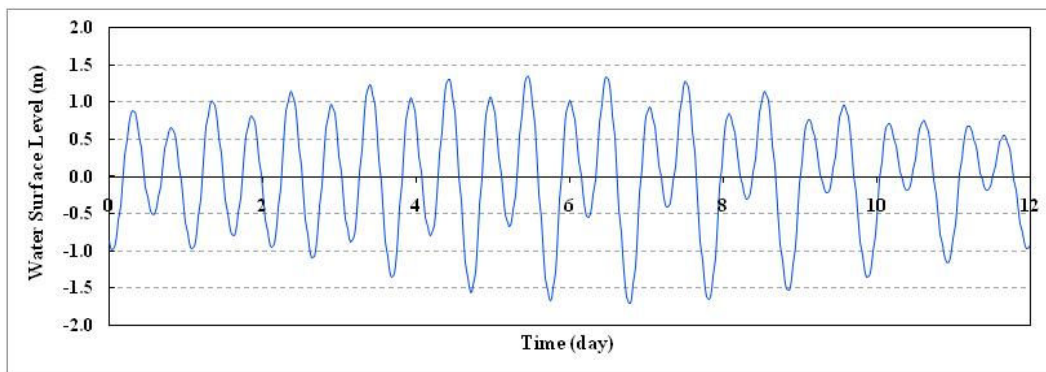

Fig. 8 Measured water surface level at Hengam Island, Hormuz Strait
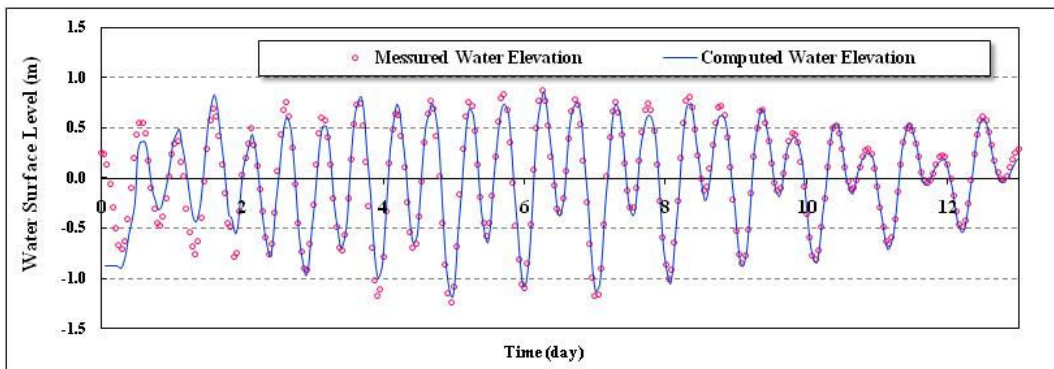

Fig. 10 Comparison of the Computed water surface level with measured data at Assaluye Port
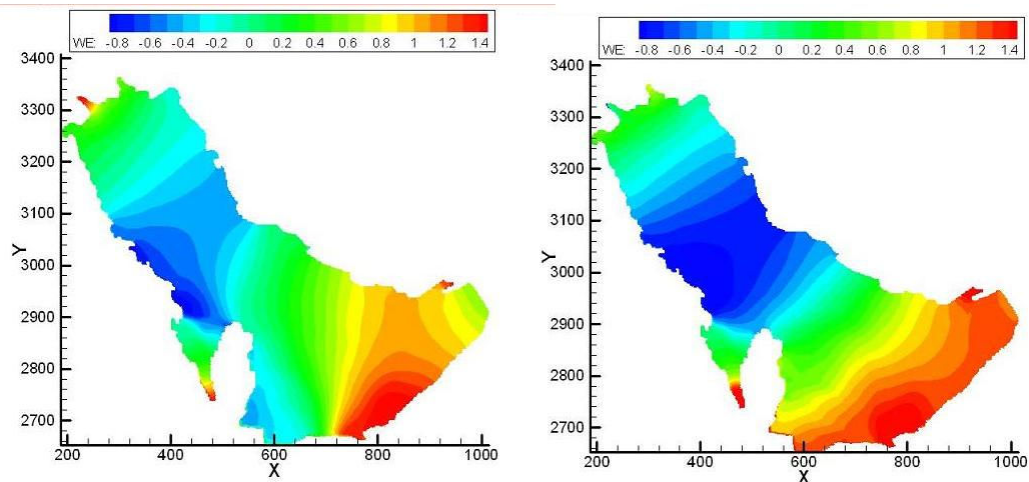
Fig. 9 Computed results of water surface elevation (a) after 132 hours (b) after 192 hours

VIII. CONCLUSION

In this paper, a numerical hydrodynamic model for the simulation of tidal currents in Persian Gulf has been introduced. The proposed governing equations for the model, the average two-dimensional shallow water equations in depth are the main techniques to solve these equations based on ADI. Due to the diversity in equations terms and to solve more precise terms, Time Splitting technique has been used without concerning a lack of congruencies of the main method for all the terms to allow the use of separate methods for each term and select the appropriate ones. Models can both be implemented in different environments with simple geometry (experimental samples) and complex geometry (coastal and marine environments with complex Bathymetry).

To ensure the accuracy of the results of this mathematical model, the results of its implementation with a number of experimental samples with analytical solutions or measured values were compared and the experimental samples of dam break has been selected for presentation in this paper. Evaluating the results show that based on the acceptability measures the model can be invoked and can be selected to be run for real marine environments.

Hydrodynamic models is actually part of the study conducted to simulate the of oil spill pollution in the Persian Gulf, so the results of simulated tidal currents in the Persian Gulf is presented by the model. The results of computational models with measured data were compared with the results in Assaluye port which show the high capabilities of this simulation model.

REFERENCES

[1] R. Garcia, and R. Kahawitha, Numerical solution of the St. Venant equations with the Mac Cormack finite difference scheme, Int. J. Numer. Method. Fluid., vol. 6, pp 507-527, 1986.
[2] R. J. Fennema, and M. H. Chaudhry, "Explicit methods for 2D transient free-surface flows," J. of Hydraul. Eng., vol. 116, pp 1013-1034, 1990.
[3] C. Bellos, J. Soulis, and J. Sakkas, "Computation of two dimensional dam break induced flow," Advances in Water Res., vol. 14, pp. 31-41, 1991.
[4] A. Akanbi, and N. Katopodes, "Model for flood propagation on initially dry land, Journal of Hydraulic Engineering," vol. 114, pp. 689-706, 1988.
[5] D. H. Zhao, H. W. Shen, G. Q. Tabios, J. S. Lai, and W. Y. Tan, "Finite-volume two dimensional unsteady-flow model for river basins," J. Hydraul. Eng., vol. 120, pp. 863-883, 1994.
[6] A. Valiani, V. Caleffi, and A. Zanni, "Case Study: Malpasset dam-break simulation using a two-dimensional finite volume method," J. Hydraul. Eng., ASCE, vol. 128, no. 5, pp. 460-472, 2002.
[7] C. Hirsch, *Numerical computation of internal and external flows*, Wiley J. and Sons, 1990.
[8] X. D. Yang, H. Y. Ma, and Y. N. Huang, "Prediction of homogeneous shear flow and a backward-facing step flow with some linear and non-linear K-g turbulence models, Communications in Nonlinear Science and numerical Simulation, vol. 10, pp. 315-328, 2005.
[9] D. Peaceman, and M. Rachford, "The numerical solution of parabolic and elliptic differential equations," J. SIAM, vol. 3, pp. 28-41, 1955.
[10] J. Douglas, and H. Rachford, "On the numerical solution of the heat conduction problem in two and three space variables," Trans. Amer. Math. pp. 421-439, 1956.
[11] J. Douglass, and J. Gunn, "A general formulation of alternating direction methods," Numer. Math., vol. 6, pp. 428, 1964.
[12] M. B. Abbott, and A. W. Minns, *Computational Hydraulics*. Ashgate Publishing Ltd, Aldershot, UK, 1997.
[13] R. D. Richtmyer, and K. W. Morton, *Difference methods for initial-value problems*, second edition. Wiley-Interscience, 1967.
[14] W. Rodi, "Turbulent Simulation in Hydraulics and Large Eddy Simulation," Advances Seminar on Education of Water Science, Sichuan University, pp. 16-21 Sep. 2001.
[15] J. Smagorinsky, "Problems and promises of deterministic extended range forecasting," Bull. Amer. Meteorol. Soc., vol. 50, No. 5, pp. 286-311, 1969.
[16] S. Murakami, "Comparison of various turbulence models applied to a bluff body," J. Wind Eng. Ind. Aerodyn., vol. 46&47, pp. 21-36, 1993.

[17] M. M. Namin, "A Fully Three-Dimensional Non-Hydrostatic Free Surface Flow Model for Hydro-Environmental Predictions", Phd Thesis, Cardiff University, 2004.

[18] J. E. Fromm, "A method of reducing dispersion in convective difference scheme," J. Comput. Phys., vol. 3, pp. 176–84, 1968.

[19] L. C. Walstra, L. C. Van Rijn, H. Blogg, and M. Van Ormondt, "Evaluation of a Hydrodynamic Area Model Based on the COAST3D Data at Teignmouth 1999," Report TR121-EC MAST Project No. MAS3-CT97-0086, HR Wallinford, UK, D4.1-D4.4, 2001.

[20] J. Sutherland, D.J.R. Walstra and H. Southgate, "Evaluation of coastal area modeling systems at an estuary mouth," Coastal Eng., vol. 51, no. 2, pp. 119–142, April 2004.

[21] J. Sutherland, A. H. Peet, and R. T. Soulsby, "Evaluation the performance of morphological models," Coast. Eng., vol. 51, pp. 917–939, 2004.

[22] E. H. L. Fernandes, K. R. Dyer, and L. F. H. Niencheski, "TELEMAC-2D calibration and validation to the hydrodynamics of the Patos Lagoon (Brazil)," J. Coast. Res., vol. 34, pp. 470-488, 2001.