Proceedings of
the 2nd ASEAN Civil Engineering Conference

sponsored by

ASEAN FOUNDATION

AUN/SEED-Net JICA

Organized by

Department of Civil Engineering
Chulalongkorn University

Department of Civil Engineering
National University of Laos

Vientiane, Lao PDR
March 11-12, 2010

ORGANIZING COMMITTEES

Organizing Committee, Chulalongkorn University

Assoc. Prof. Dr. Boonsom Lerdhirunwong	Dean, Faculty of Engineering
Assoc. Prof. Dr. Phoonsak Pheinsusom	Head, Department of Civil Engineering
Assoc. Prof. Dr. Tirawat Boonyatee	
Assist. Prof. Dr. Veerasak Likhitruangsilp	
Assist. Prof. Dr. Vachara Peansupap	
Assist. Prof. Dr. Tanate Srisirirojanakorn	
Assist. Prof. Dr. Kasem Choocharukul	
Assist. Prof. Dr. Manoj Lohatepanont	
Assist. Prof. Dr. Suched Likitlersuang	
Assist. Prof. Dr. Noppadon Jokkaw	
Assist. Prof. Dr. Saksith Chalermpong	
Assist. Prof. Dr. Jaroon Rungamornrat	
Dr. Boonchai Sangpetngam	
Dr. Akhrawat Lenwari	
Dr. Jittichai Rudjanakanoknad	
Dr. Withit Pansuk	

Organizing Committee, National University of Laos

Assoc. Prof. Dr. Boualinh Soysouvanh	Dean, Faculty of Engineering
Assoc. Prof. Nhinxay Visane	Head, Department of Civil Engineering
Dr. Khampaseuth Thepvongsa	
Ms. Pipong Phimphachan	

International Technical Committee

Muhamad Abduh, Institute of Technology Bandung, Indonesia
Hamidi Abdul Aziz, Universiti Sains Malaysia, Malaysia
Chhouk Chhay Horng, Institute of Technology of Cambodia, Cambodia
Ronaldo S. Gallardo, De la Salle University, Philippines
Bambang Ismanto Siswosoebroto, Institute of Technology Bandung, Indonesia
Seiichi KAGAYA, Hokkaido University, Japan
Panitan Lukkunaprasit, Chulalongkorn University, Thailand
Phoonsak Pheinsusom, Chulalongkorn University, Thailand
Hamzah A. Rahman, University of Malaya
Guillermo Q. Tabios III, University of the Philippines Manila, Philippines
Shinei Takano, Hokkaido Univeristy, Japan
Hiroyuki TANAKA, Hokkaido University, Japan
Bui Cong Thanh, Ho Chi Minh City University of Technology, Vietnam
Wanchai Teparaksa, Chulalongkorn University, Thailand
Nhinxay Visane, National University of Laos, Lao People's Democratic Republic
Arnon Wongkaew, Burapha University, Thailand
Siam Yimsiri, Burapha University, Thailand
Mohd Zamin Bin Jumaat, University of Malaya, Malaysia

TABLE OF CONTENTS

Optimal Time Splitting For Two-Dimensional Depth Averaged Flow Simulation

Muhammad CAHYONO
Associate Professor
Department of Civil Engineering
Faculty of Civil and Environmental Engineering
Institut Teknologi Bandung 40132
Jl. Ganesha 10, Bandung
Fax: +062-022-2504987
E-mail: mcahyono2003@yahoo.co.uk

Yulius EKA SAPUTRA
Research Associate
Engineering Center for Industry
Institut Teknologi Bandung
Jl. Ganesha 10, Bandung 40132
Fax: +062-022-2504987

Abstract : This paper describes the use of the optimal time-splitting technique in the development of a two-dimensional depth-average hydrodynamic model to simulate the flow in two-dimensional plane. The hydrodynamic equations utilized in the model consist of depth integrated equations of continuity and momentum in two dimensional horizontal plans. The effecs of wind stress and Coriolis force as well as the effecs of radiation stress due to wave breaking are considered in the momentum equations. In this technique, the hydrodynamic equations are split into successive one-dimensional problems, including momentum advections, wave propagations including momentum sources and sinks due to bed friction, surface wind stress, Coriolis force and radiation stress, and horizontal momentum diffusions. The momentum advections are solved by using the method of characteristic with shape-preserving piecewise cubic spline being used for interpolation in order to produce free oscillation solutions in region where high gradients of velocities present, particularly in area where the effects of advective acceleration terms in the momentum equations are significant. The wave propagations are solved using the MacCormack scheme, while the horizontal momentum diffusions are solved using the second-order central scheme.

The splitting technique is optimal in the sense that the Courant number constraint is reduced to be only dependent upon the constraint for the one-dimensional scheme and that the most suitable scheme can be applied for solving each sub-problem. Various tests have been selected to assess the performance of the technique has been performed and the results are reported. The model has also been applied to simulate the flows due to combined actions of tide and wind waves around the Cilacap Fishing Port.

Keywords: *two-dimensioanal flow, optimal time splitting, method of characteristic, shape-preserving piecewise cubic interpolation*

1. INRODUCTION

Several depth-averaged numerical models have been developed and applied to free-surface water problems. The depth-averaged model may produce reasonable results in many flow simulations for practical engineering applications. However, for convection-dominates flow, such as circulated flow, and sock capturing problems such as hydraulic jump and dam-break flows, or flow through rapidly varied channels, numerical difficulties and unsatisfactory simulated flow patterns may occur. Benque *et al.* (1982) proposed a fractional three split operator approach to overcome difficulties in dealing with circulation flow simulations. The method they proposed splits the hydrodynamic equations into three step problems, namely advection, diffusion and wave propagation. The method of characteristics was used to solve the advection, with convection being treated separately in each spatial direction in order to produce high accurate solutions for the convection. The diffusion was calculated using an implicit finite difference scheme, and wave propagation was calculated using an alternating direction implicit (ADI) scheme. Using a similar approach, Lien *et al.* (1999) developed a two-step split-operator solution procedure for solving the shallow water flow equation. The procedure involves two steps. The first step was to compute the velocity in the momentum equation without the pressure gradient and the second step was to solve wave propagation including the effects of the pressure gradient and bed friction. The circulation flow in circular basin has been selected to validate the model. Moreover, Hsu *et al.* (2000) used the explicit-finite-analytic method with second order accuracy in space and first order accuracy in time for treatment the convective terms, while other terms of the momentum equation were discretized by the finite difference method. They concluded that the model can compute the convection-dominated flow with strong curvature of streamline, hydraulic jump with presence of sub-critical and supercritical flow regimes simultaneously, and unsteady dam-break flows in the non-prismatic channel.

Many authors such as Bellos and Sakas (1987), Glaister (1988), Fennema and Chaudhry (1990), Aureli *et al.* (2008) and Erpicum *et al.* (2009) have developed the 1-D and 2-D flow models for hydraulic jump and dam break flow numerical computations. The various techniques have been developed to solve 1-D and 2-D flow equations. Most of them adopted the shock capturing techniques, such as flux vector splitting and TVD scheme for capturing jump or discontinuities.

Abarbanel and Gottlieb (1981) proposed a splitting technique, called "optimal time splitting" for solving the Navier-Stokes equation. The algorithm proposed by them was higher-order accurate in space and time. With this technique the multi-dimensional problem was split into successive one-dimensional problems and each one-dimensional problem was then solved using the most suitable finite difference scheme. Hence, the splitting technique was optimal in the sense that the Courant number constraint was reduced to be only dependent upon the constraint for the one-dimensional scheme. Cahyono and Falconer (1997) use this splitting technique to solve multi-dimensional scalar transport. The ULTIMATE QUICKEST scheme of Leonard (1991) has been used in the algorithm for solving the pure convections. The diffusion equation was solved by using the second order implicit centred scheme, while the reaction equation can be solved by using the the Runge-Kuta method to get higher order accurate solution in time. Three test cases have been considered to gain insight the merit of the algorithm, including advection of sharp discontinuity by steady state velocity field in a square region, pure advection of a Gaussian scalar distribution for rotating fluid around a square of side length of 2, and two-dimensional advective-diffusion equation for a passive scalar in rectangular region. The resulting tests show that the algorithm is computationally efficient and accurate when used for multi-dimensional advective-diffusion simulations.

This paper describes the use of the optimal time-splitting technique in the development of a two-dimensional depth-average hydrodynamic model to simulate the flow in two-dimensional plane. The technique is based on the optimal time splitting proposed by Abarbanel and Gottlieb (1981). In this technique the hydrodynamic equations are split into successive one-dimensional problems, including momentum advections, wave propagations including momentum sources and sinks due to bed friction, surface wind stress, Coriolis force and radiation stress due to wind wave effects, and horizontal momentum diffusions. The higher-order non-oscillatory scheme such as Total Variation Diminishing (TVD) schemes, universal limiter (ULTIMATE) schemes, Godunov type schemes, the quasi-monotone schemes, characteristic type schemes with shape-preserving interpolation functions and TVD filter algorithms may be used to solve the 1-D momentum advection equation. In this model the 1-D momentum advections are solved by using the method of characteristic with shape-preserving piecewise cubic spline being used for interpolation in order to produce highly accurate free-oscillation solutions in region where high gradients of velocities present, particularly in area where the effects of advective acceleration terms in the momentum equations are significant. The wave propagations are solved using the MacCormack scheme. The TVD filter or TVD diffusion may be incorporated into the MacCormack scheme when dealing with shock capturing problems such as hydraulic jump and dam break flow computations. The horizontal momentum diffusions are solved using the second-order central scheme. The proposed model can be used to compute the convection-dominated flow, hydraulic jump, transitional flow and dam-break flows. However, this paper aims on the flow computation with strong circulation. For this purpose, three test cases have been considered for model testing, including flow in a circular basin, flow around a breakwater and flow around a jetty. The model has also been applied to simulate the characteristics of flow due to combined actions of tide and wind waves around the Cilacap Fishing Port.

2. GOVERGING EQUATIONS

The governing equations for a 2D shallow water flow in Cartesian co-ordinates may be written as:

$$\frac{\partial \eta}{\partial t} + \frac{\partial (h+\eta)U}{\partial x} + \frac{\partial (h+\eta)V}{\partial y} = 0, \tag{1}$$

$$\frac{\partial U}{\partial t} + U\frac{\partial U}{\partial x} + V\frac{\partial U}{\partial y} + g\frac{\partial \eta}{\partial x} + F_x - D_x + R_x + C_x + W_x = 0, \tag{2}$$

$$\frac{\partial V}{\partial t} + U\frac{\partial V}{\partial x} + V\frac{\partial V}{\partial y} + g\frac{\partial \eta}{\partial y} + F_y - D_y + R_y + C_y + W_y = 0, \tag{3}$$

where x and y represent the Cartesian co-ordinates in the horizontal plane; U and V are the depth-averaged velocity components in the x and y-directions; h is the water depth; η is water level above the datum, F_x and F_y are the bed stress components in the x and y-directions, W_x and W_y are the wind stress components in the x and y-directions, R_x and R_y are the radiation stress components due wave action in the x and y-directions, C_x and C_y are the Coriolis force components in the x and y-directions and D_x and D_y are the dispersion terms.

F_x and F_y are described by the depth-averaged velocities and wave velocities as

$$F_x = \frac{C_f}{h+\eta}(U+u_b)\sqrt{(U+u_b)^2 + (V+v_b)^2}, \tag{4}$$

$$F_y = \frac{C_f}{h+\eta}(V+v_b)\sqrt{(U+u_b)^2 + (V+v_b)^2}, \tag{5}$$

where C_f is a bed shear stress coefficient relating to combined effects of current and wave, u_b and v_b are the wave driven bed stress components in the x and y-directions. Details information for obtaining , u_b and v_b can be seen in van Rijn (1993).

R_x and R_y can be computed from the following equations, i.e.

$$R_x = \frac{1}{\rho(h+\eta)}\left(\frac{\partial S_{xx}}{\partial x} + \frac{\partial S_{xy}}{\partial y}\right), \tag{6}$$

$$R_y = \frac{1}{\rho(h+\eta)}\left(\frac{\partial S_{xy}}{\partial x} + \frac{\partial S_{yy}}{\partial y}\right), \tag{7}$$

where ρ is the density of water, S_{xx}, S_{xy} and S_{yy} are Radiation stress components defined by

$$S_{xx} = \varepsilon\frac{C_g}{C}\frac{k_1k_1}{k^2} + \varepsilon\left(\frac{C_g}{C} - \frac{1}{2}\right), \tag{8}$$

$$S_{xy} = \varepsilon\frac{C_g}{C}\frac{k_1k_2}{k^2}, \tag{9}$$

$$S_{yy} = \varepsilon\frac{C_g}{C}\frac{k_2k_2}{k^2} + \varepsilon\left(\frac{C_g}{C} - \frac{1}{2}\right), \tag{10}$$

C is wave celerity, C_g is group velocity of wave, k = wave number, $k_1 = k\cos\alpha$, $k_2 = k\sin\alpha$ and ε wave energy given by

$$\varepsilon = \frac{\rho g H^2}{8}. \tag{11}$$

Dispersion terms, D_x and D_y due to turbulent and non-uniformities velocities are expressed as

$$D_x = \frac{\partial}{\partial x}\left(\varepsilon_{xx}\frac{\partial U}{\partial x}\right) + \frac{\partial}{\partial y}\left(\varepsilon_{xy}\frac{\partial U}{\partial y}\right), \tag{12}$$

$$D_y = \frac{\partial}{\partial x}\left(\varepsilon_{yx}\frac{\partial V}{\partial x}\right) + \frac{\partial}{\partial y}\left(\varepsilon_{yy}\frac{\partial V}{\partial y}\right), \tag{13}$$

where $\varepsilon_{xx} = \varepsilon_{xy} = \varepsilon_{yx} = \varepsilon_{yy} = \varepsilon$ are horizontal dispersion coefficients given by Equation (20).

$$\varepsilon = Nl\sqrt{g(h+\eta)}, \tag{14}$$

where N and l are empirical coefficients.

3. NUMERICAL SOLUTION

3.1 Numerical Algorithm

The Equations (1) to (3) are solved by using an optimal time splitting, a technique similar with that proposed by Abarbanel and Gottlieb (1981). Using this splitting technique, the governing hydrodynamic equations (1) to (3) are split into successive one-dimensional problems by the following procedure:

$$\Phi^{n+2} = L_{Cx}L_{Cy}L_{Px}L_{Py}L_{Dx}L_{Dy}L_{Dy}L_{Dx}L_{Py}L_{Px}L_{Cy}L_{Cx}\Phi^n, \tag{15}$$

where $\Phi = \{\eta, U, V\}^T$;

L_{Cx} and L_{Cy} are finite difference schemes that solve the 1-D momentum convection equations in the x and y-directions respectively, i.e.

$$\frac{\partial U}{\partial t} + U\frac{\partial U}{\partial x} = 0, \tag{16}$$

$$\frac{\partial V}{\partial t}+U\frac{\partial V}{\partial x}=0\,; \tag{17}$$

$$\frac{\partial U}{\partial t}+V\frac{\partial U}{\partial y}=0\,, \tag{18}$$

$$\frac{\partial V}{\partial t}+V\frac{\partial V}{\partial y}=0\,, \tag{19}$$

L_{Px} and L_{Py} are finite difference schemes that solve the 1-D wave propagation and momentum source and sinks due to bed friction, surface wind stress, Coriolis force and Radiation stress in the x and y-directions respectively, i.e.

$$\frac{\partial \eta}{\partial t}+\frac{\partial (h+\eta)U}{\partial x}=0\,, \tag{20}$$

$$\frac{\partial U}{\partial t}+g\frac{\partial \eta}{\partial x}+F_x+R_x+C_x+W_x=0\,, \tag{21}$$

$$\frac{\partial \eta}{\partial t}+\frac{\partial (h+\eta)U}{\partial y}=0\,, \tag{22}$$

$$\frac{\partial U}{\partial t}+g\frac{\partial \eta}{\partial y}+F_y+R_y+C_y+W_y=0\,; \tag{23}$$

L_{Dx} and L_{Dy} are finite difference scheme that solve the 1-D horizontal momentum dispersion in the x and y-directions respectively, i.e.

$$\frac{\partial U}{\partial t}=\varepsilon\frac{\partial^2 U}{\partial x^2}\,, \tag{24}$$

$$\frac{\partial V}{\partial t}=\varepsilon\frac{\partial^2 V}{\partial x^2}\,, \tag{25}$$

$$\frac{\partial U}{\partial t}=\varepsilon\frac{\partial^2 U}{\partial y^2}\,, \tag{26}$$

$$\frac{\partial V}{\partial t}=\varepsilon\frac{\partial^2 V}{\partial y^2}\,; \tag{27}$$

The 1-D equations (16) to (27) can then be solved by using the most suitable finite difference scheme and then the Courant number constraint of the algorithm given by Equation (15) was dependent upon the constraint for the one-dimensional scheme. In this model, the 1-D momentum convection equations (16) to (19) are solved using a characteristic method with shape-preserving piecewise cubic spline being used for interpolation in order to produce highly accurate free-oscillation solutions in region where high gradients of velocities present. The 1-D wave propagation equations (20) to (23) are solved using the MacCormack scheme. When shock phenomenon are present in the flow such as hydraulic jump and dam failure flows, the TVD filter or TVD diffusion may be incorporated into the MacCormack scheme to suppress the oscillations around the jump or discontinuities. The horizontal momentum diffusion equations (24) to (27) are solved by using the second-order explicit or implicit central schemes.

3.2 Boundary Condition

The velocity field or discharge and water level fluctuation may always be specified as an inflow and outflow boundary conditions. For solid boundary, free slip boundary assumption and the method of characteristic are used in L_{Cx} and L_{Cy} for velocity computations, whereas forward and backward differences are used in L_{Px} and L_{Dx} or wave propagation computations.

4. MODEL TESTS AND APPLICATION

4.1 Steady circulation in circular basin

This test is chosen to verify the capability of the model to predict the flow circulation due to the influence of advection effect in the circular basin flow. The experiment and simulation test were initially carried out by Falconer (1976) and then the test was usually used by others authors to perform their model to simulates flow circulation, including Benque *at al.* (1982), Lien *at al.* (1999) and Hsu *et al.* (2000). Geometry of the circular basin is circular, with vertical sidewalls. Difference with the Falconer's test, in this test a flat bed channel is setup in the middle of the basin. The bottom of channel is made lower than that of basin. Three conditions are considered in this test. The first is simulation with a uniform channel, the second and third tests are simulations for narrow approach and narrow exit channels. The width of the inlet for uniform channel is 0.08 m, width of inlet or outlet is 0.24 m, and the diameter of the circle is 1.5 m. The geometries of the circular basins for tests considered are shown in Figure 1.

The simulations were carried out for a steady discharge of 0.02 m^3/s. Figure 2 shows the result for the circulatory flow patterns with in the circular basin for uniform channel, where Figure 3 and 4 are results of simulations for narrow inlet and narrow outlet channels. As can be seen, the model can predict circulation flow for three cases considered. To see the important effect of the advective acceleration terms in generating the circulation flow the model was also be run without the advective acceleration terms. The results of simulation for the uniform channel case are shown in Figure 5. It shows that the model produces no circulation flow.

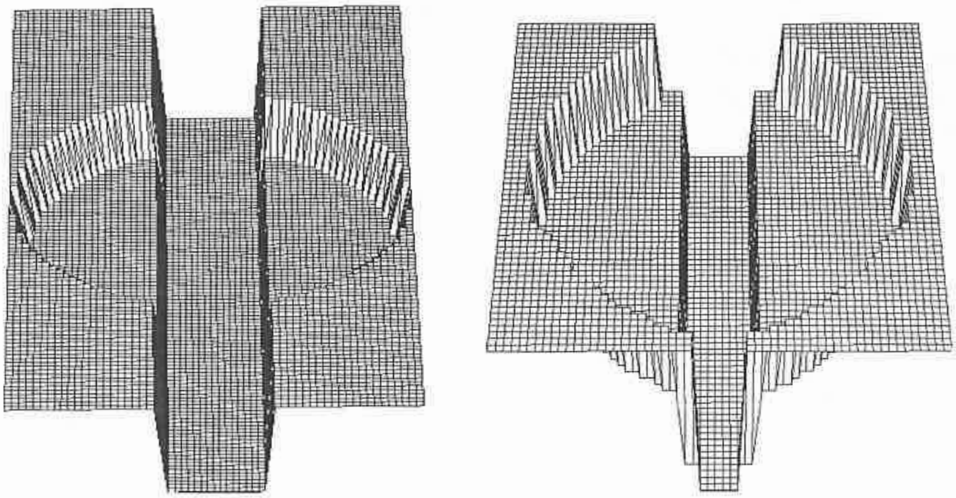

Figure 1 Circular basin. (a) uniform width channel and (b) narrow inlet or outlet channel

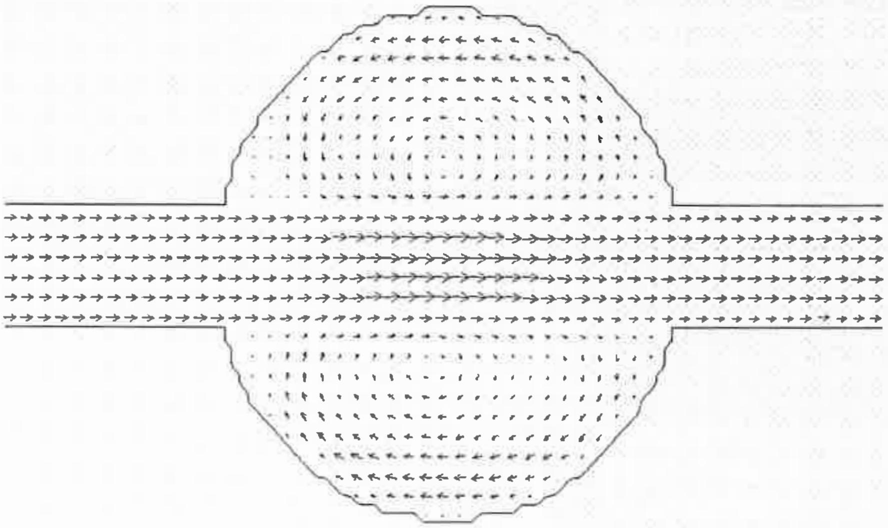

Figure 2 Steady circulation in circular basin for uniform channel case

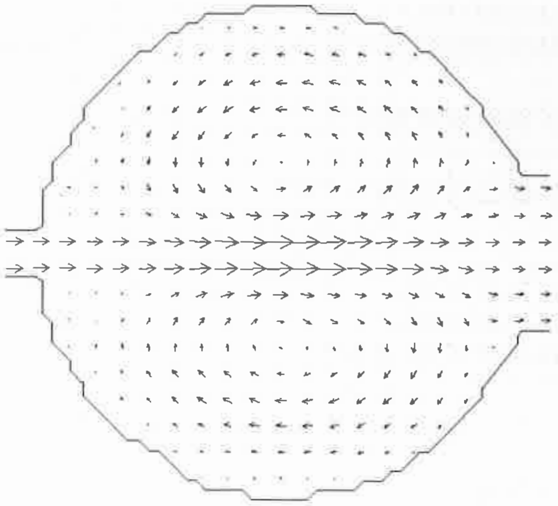

Figure 3 Steady circulation in circular basin for narrow inlet channel case

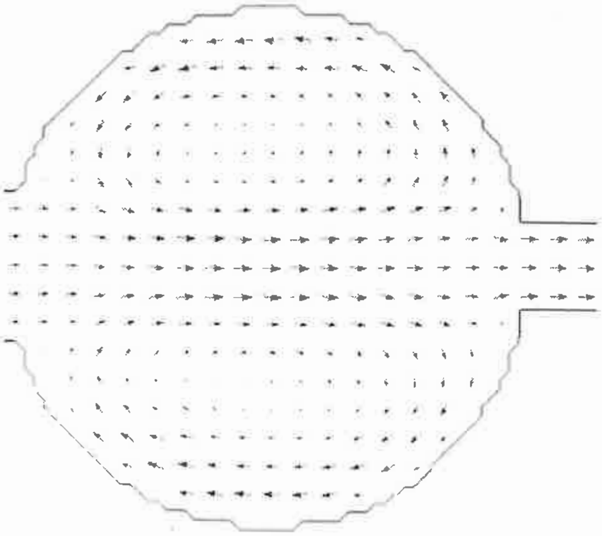

Figure 4 Steady circulation in circular basin for narrow outlet channel

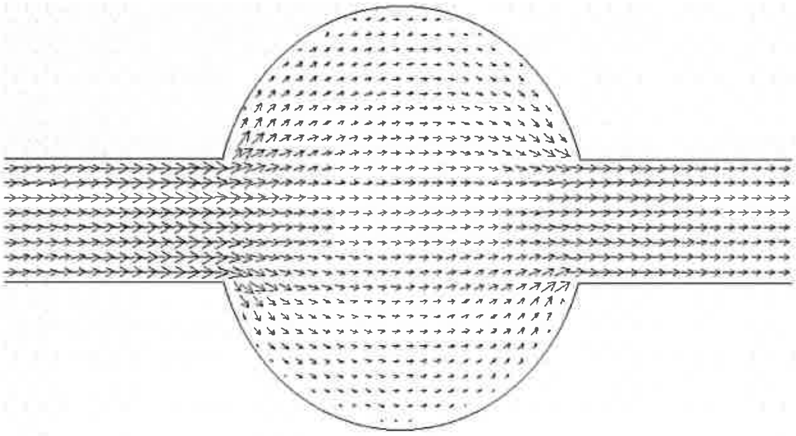

Figure 5 Flow in circular basin simulated without momentum advection terms

4.2 Circulation behind breakwater

This second test is performed to predict the flow circulation behind a breakwater where the zone of flow separation exists. Simulations were carried out for steady flow condition with constant velocities being prescribed at inflow and fixed water levels being setup at upstream boundary. Figure 6 and 7 show the velocity field simulated by the model. As can be seen the circulation flow is well predicted behind the breakwater, in the zone of separation flow. In order to see the effect of convective terms in generating circulation, another simulation without the convective terms has also been carried out. The simulation results can be seen in Figure 8. No circulation flow presents in the velocity field. It shows that the convective terms have strongly influence in generating circulation flow.

4.3 Wave induced circulation around jetty

Wave induced circulation in refraction-diffraction and breaking zone has been simulated. The geometry of jetty and bathymetry coastal area being simulated is given in Figure 9. Bottom slope is 1:84. Simulations were carried out for wave height of 1m, period of 5 second and direction of wave to the coastline line being set to 45^0. Figure 10 and Figure 11 show the wave-driven flow around a transversal jetty simulated by the model. It shows that the model can predict circulation behind the jetty where separation zone flow presents.

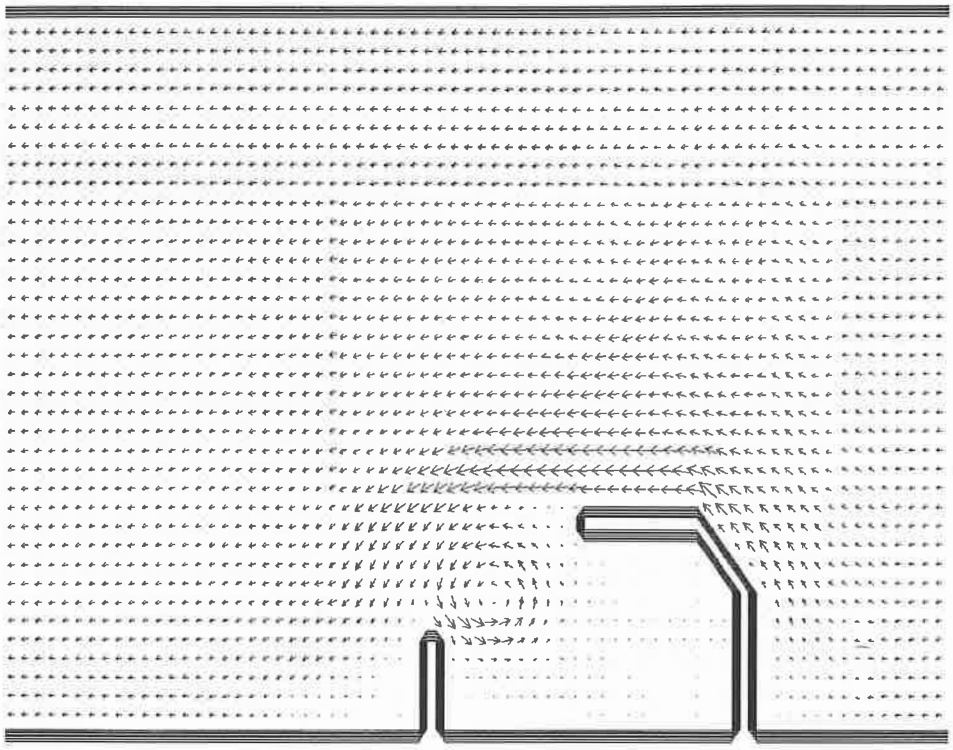

Figure 6 Circulation on region of separation flow around breakwater

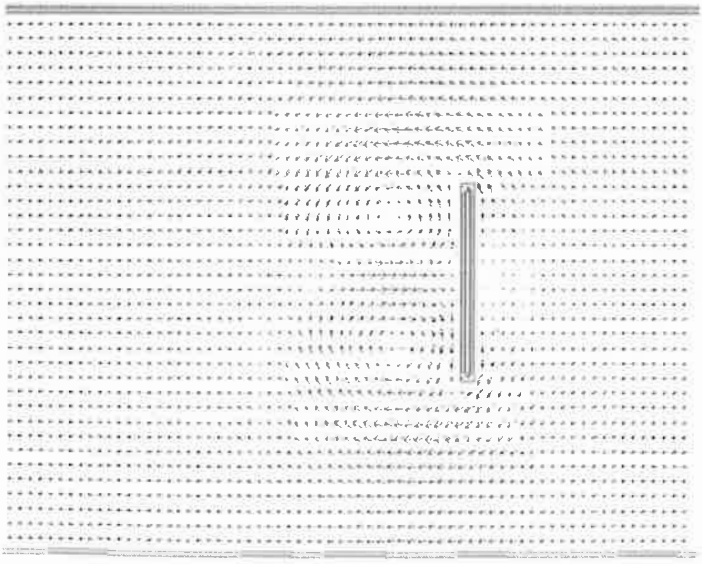

Figure 7 Circulation on region of separation flow behind breakwater

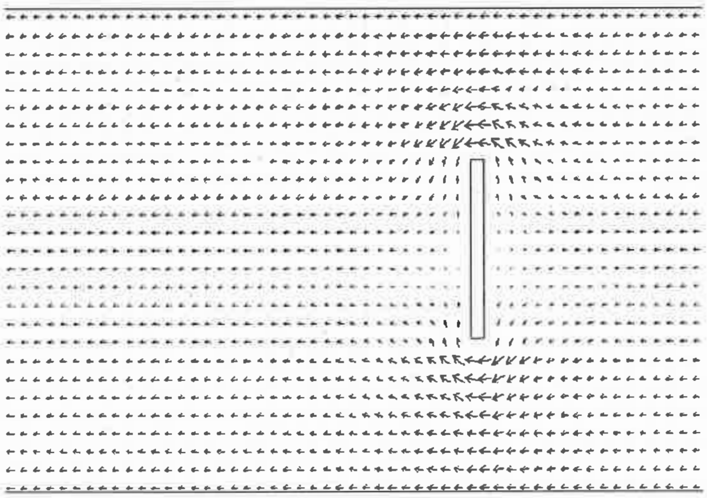

Figure 8 Flow around breakwater simulated without momentum advection terms

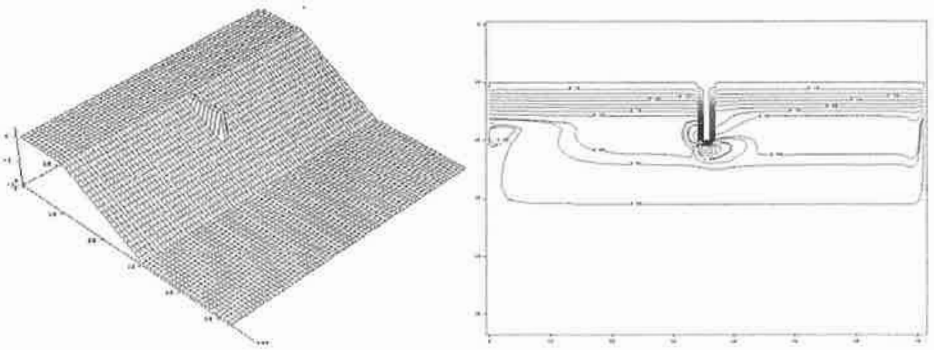

Figure 9 (a) Model grid and (b) bathymetry of the coastal area

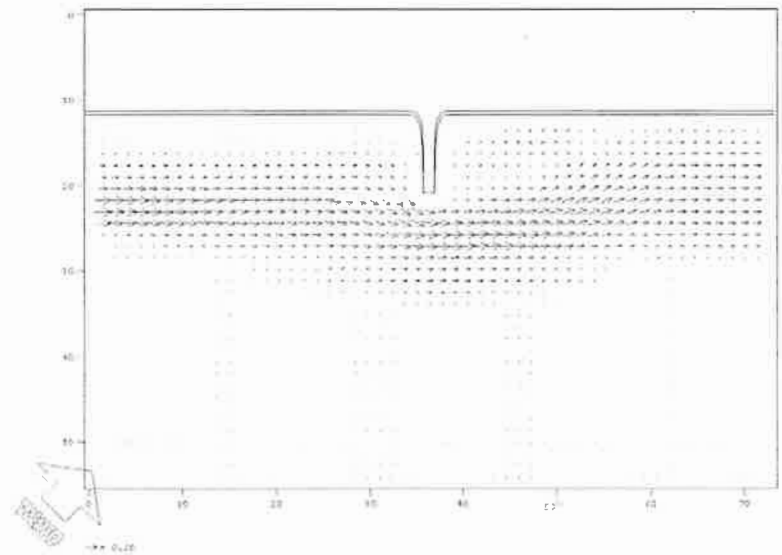

Figure 9 Wave induce flow around jetty

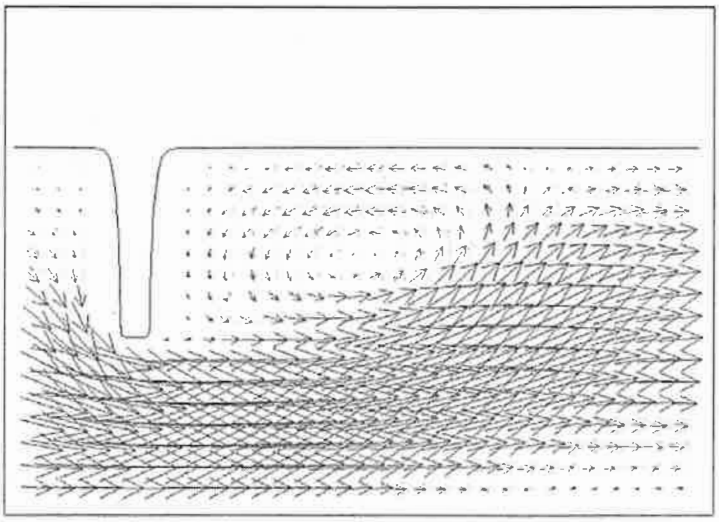

Figure 10 wave induce circulation behind jetty

4.4 Model Application

The model has also been applied to simulate the flows due to combined actions of tide and wind waves around the Cilacap Fishing Port (CFP). The flow modeling study was a part of activities on the detail design project for optimization of breakwater configuration of the CFP. Figure 11 shows a bird view of the CFP taken from a helicopter while Figure12 shows the bathymetry of the coastal region around the CFP.

In applying the numerical model at the CFP, the region around the CFP Bay was represented in the model using a nested procedure consisting of coarse, mid and fine grid models. Firstly, a coarse grid model was set up covering a larger region from Pangandaran at western boundary to Kebumen at eastern boundary. This coarse grid model then provided boundary conditions for the mid grid model which cover a region smaller than that of the coarse grid model. Then, the fine model covering the region around the CFP with fine grid size can be run. Boundary conditions for the fine model were obtained from the results of the mid model simulations. In the fine model, the region around the CFP was represented in the model using grid model of 100 x 130 grid points, each of grid size $\Delta x = 8$ m, being set up. Boundary conditions used were tidal curves at western and eastern boundaries provide from the mid model simulations, streamline at southern boundary and flow velocity records at Yasa river at open northern boundary (see Figures 13).

In order to assess the influence of the wave in flow field characteristics, the simulations were carried for two conditions, namely simulations of flow due to wind wave action only and simulation of flow due to combined actions of tide and wind wave. It was noted that wind effect and Corriolis force were neglected in this simulation. Wave field around the CFP were obtained by using the W-DIVAST model. The W-DIVAST model used a nested procedure described previously to propagate wind wave from deep sea water to coastal region around the CFP. Figure 14 shows wave height field around the CFP as results of refraction-diffraction of deep sea wind wave with height of 1.5 m, period of 5 second and direction from South-East simulated by the W-DIVAST model. The model was run using a time step of 0.75 second. Figure 15 shows the velocity field due to wave action where Figure 16 is velocity field of flow due to tidal action. Combined action of wind wave and tide produce the velocity field as shown in Figure 17.

Figure 11 Bird view of Cilacap Fishing Port before extension breakwater
Source: Proyek Pengembangan Pelabuhan Perikanan Samudera Cilacap

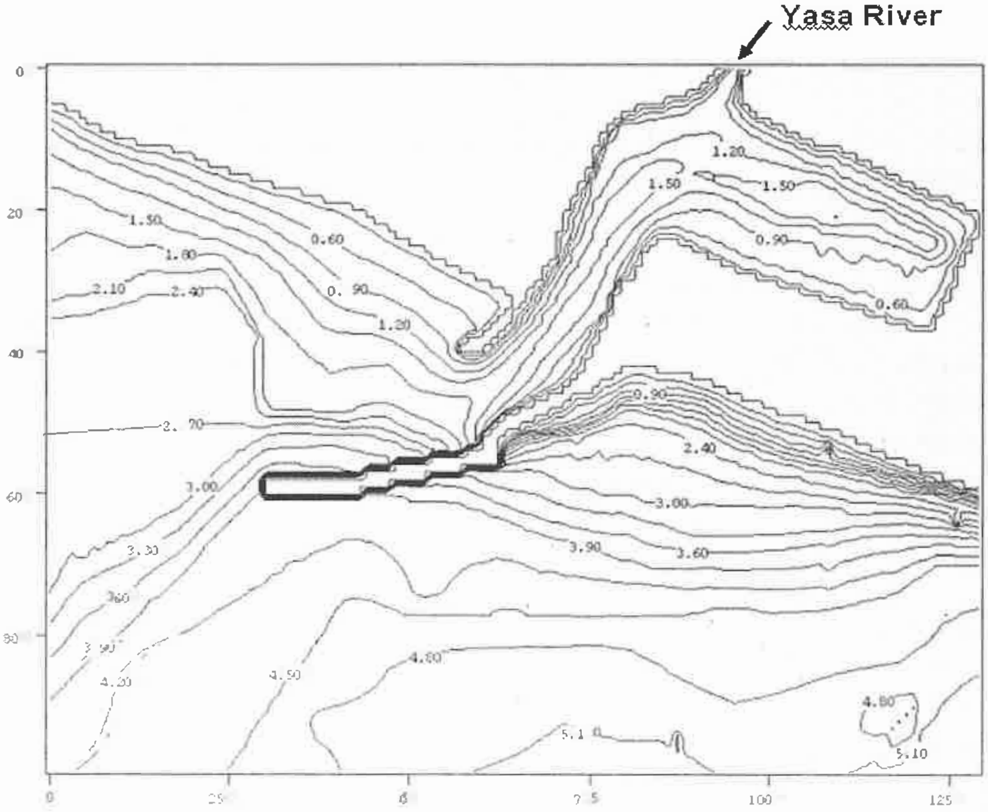

Figure 12. Bathymetry of region around Cilacap Fishing Port

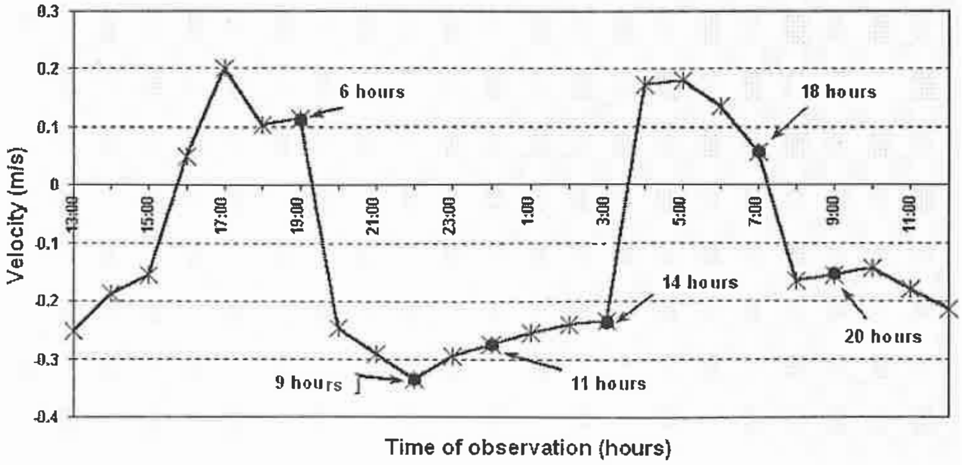

Figure 13. Velocity records at Yasa River

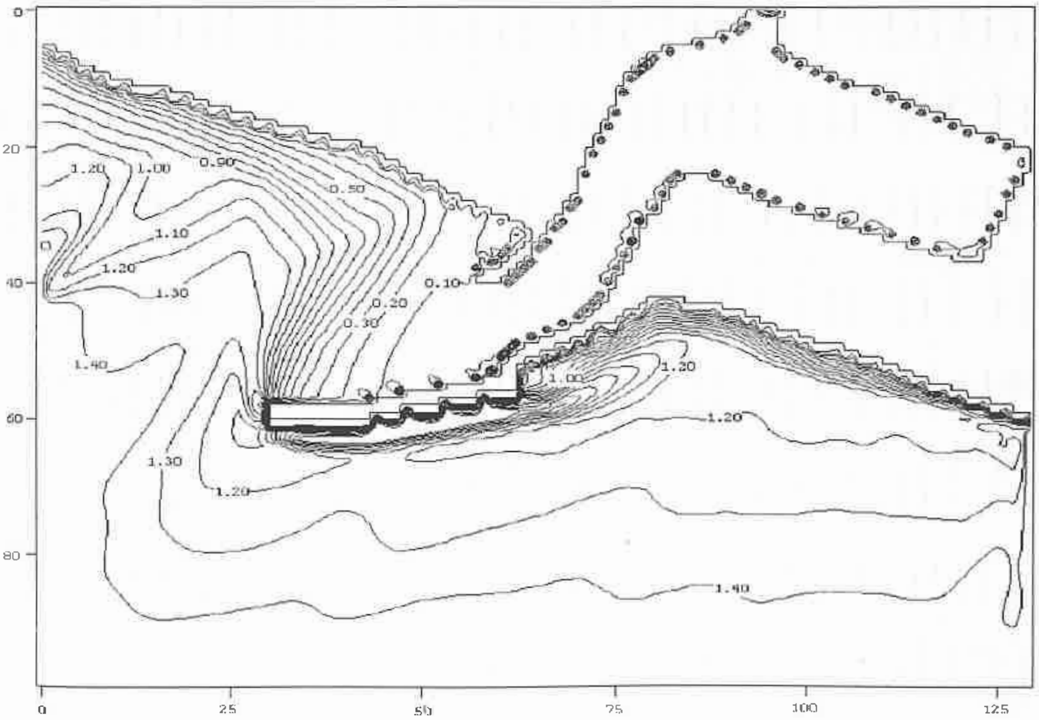

Figure 14. Wave height around Cicalap Fishing Port

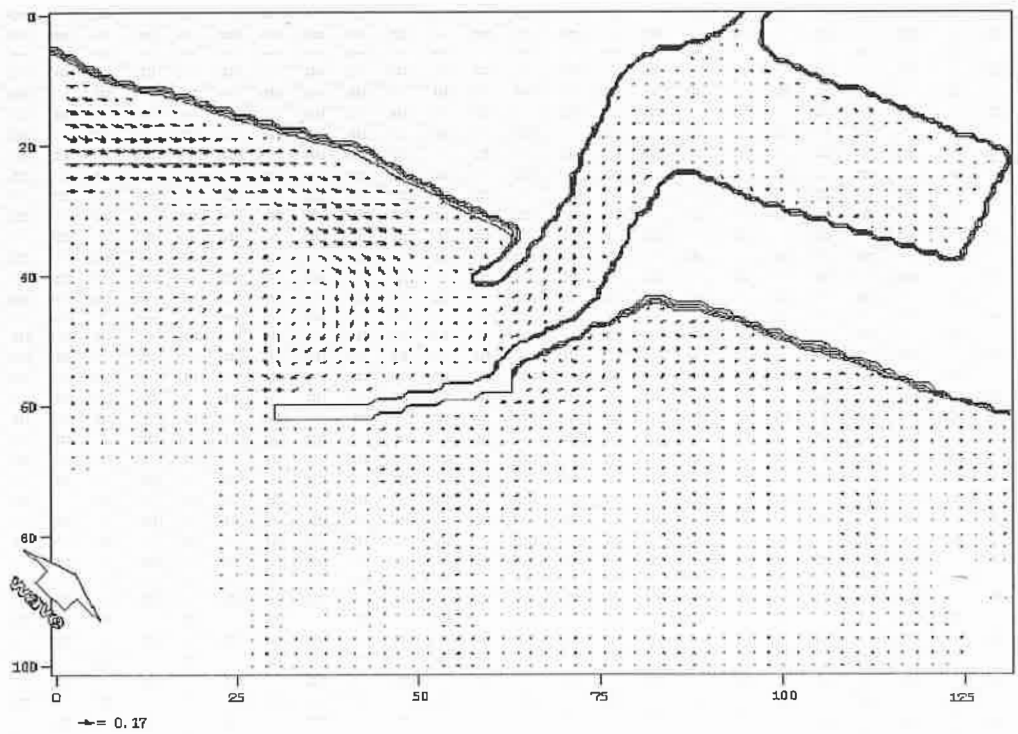

Figure 15 Velocity vector of flow due to wind wave effect around Cilacap Fishing Port

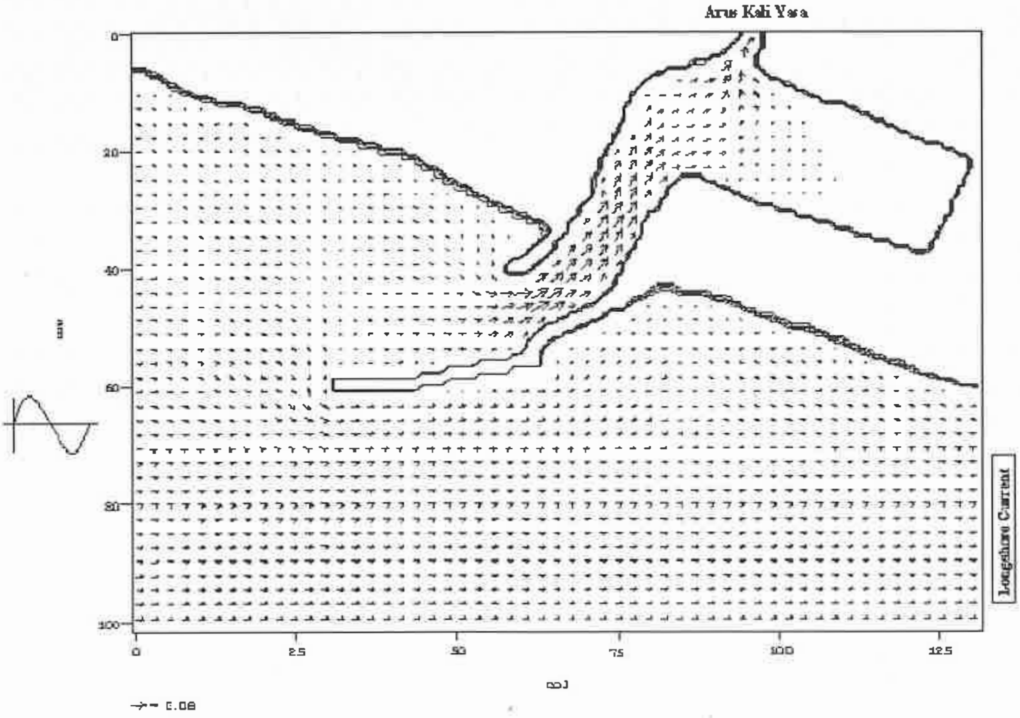

Figure 16 Velocity vector of flow due tidal action around Cicacap Fising Port

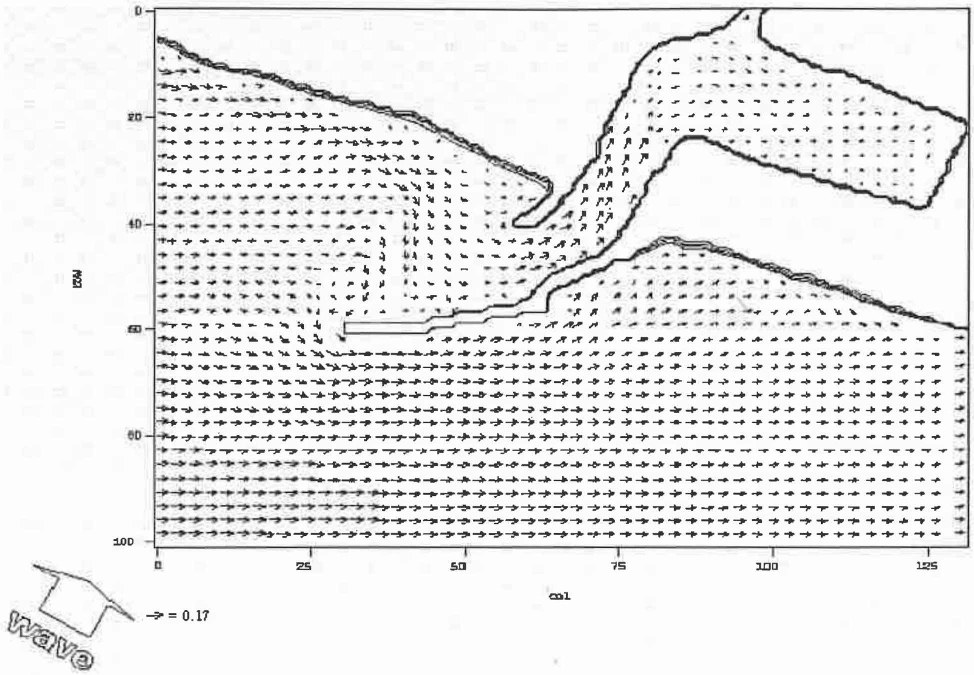

Figure 17 Velocity vector of flow due to combined actions of wind and tide around Cilacap Fising Port

5. CONCLUSIONS

A depth-averaged flow model has been developed to simulate the flow in two-dimensional plane. The governing equations are split into successive one-dimensional problems, including momentum advections, wave propagations including momentum sources and sinks due to bed friction, surface wind stress, Coriolis force and radiation stress due to wind wave effects, and horizontal momentum diffusions. The 1-D momentum advections are solved by using the method of characteristic with shape-preserving piecewise cubic spline being used for interpolation in order to produce highly accurate free-oscillation solutions. The wave propagations are solved using the MacCormack scheme. The TVD filter or TVD diffusion may be incorporated into the MacCormack scheme when dealing with shock capturing problems such as hydraulic jump and dam break flow computations. The horizontal momentum diffusions are solved using the second-order central scheme. Three test cases have been considered for model testing, including flow in a circular basin, flow around a breakwater and flow around a jetty. The model has also been applied to simulate flow due to combined actions of tide and wind waves around the Cilacap Fishing Port. The results show that the model is of capable of predicting convection-dominates flow such as circulation flows present in the zone of separation flow.

6. REFERENCES

Abarbanel, S and Gottlieb, D, (1981) Optimal Time Splitting for Two- and Three-Dimensional Navier-Stokes Equations with Mixed Derivatives, **Journal of Computational Physics, Vol. 41**, 1-33.

Aureli, F., Maranzoni, A., Mignosa, P and Ziveri, C. (2008) Dam-break flows: Acquisition of experimental data through an imaging technique and 2-D numerical modeling, **Journal of Hydraulic Engineering ASCE, Vol.134, No.8**, 1089-1101.

Bellos, C. V. and Sakkas, J. G. (1987) 1-D dam-break flow-wave propagation on dry bed, **Journal of Hydraulic Engineering ASCE Vol.113, No.12**, 1510-1524.

Benque, J. P., Cunge, J. A., Fruillet, J., Hauguel, A. and Holly, F. M. (1982) New method for tidal current computation, **Journal of the Waterway, Port, Coastal and Ocean Division, ASCE, Vol.108, No.WW3**, 396-417.

Cahyono, M. and Falconer, R. A. (1997) Optimal Time Splitting for Two- and Three-Dimensional Advective-Diffusion Simulations Using Higher Order Finite Difference Schemes, **Proc. Regional Seminar on Computational Method and Simulations in Engineering (CMSE'97)**, Institute of Technology Bandung, pp. VII.C.5.1-10.

Erpicum, S., Dewals, B. J., Archambeau, P. and Pirotton, M. (2009) Dam break flow computation based on an efficient flux vector spliiting, **Journal of Computational and Applied Mathematics.**

Falconer, R. A (1976) **Mathematical modeling of jet-forced circulation in reservoirs and harbours,** Ph.D. Thesis, Imperial College London.

Hsiu, C. T., Yeh, K. C, and Yang, J. C. (2000) Depth-averaged two-dimensional curvilinear explicit finite analytic model for open-channel flows, **Int. J. Numer. Meth. Fluids, Vol.33**, 175-202.

Fennema, R. J. and Chaudhry, M. H. (1990) Explicit methods for 2-D transient free surface flows, **Journal of Hydraulic Engineering ASCE, Vol.116, No.8**, 1013-1034.

Glaister, P (1988) Approximation Riemann solutions of the shallow water equations, **Journal of Hydraulic Research, Vol.26, No.3**, 293-300.

Leonard, B. P, (1991) The ULTIMATE Conservative Difference Scheme Applied to Unsteady One-Dimensional Advection, **Comput. Methods Appl. Mech. Engrg., Vol.88**, 17-74, 1991.

Lien, H. C., Hsieh, T. Y, and Yang, J. C. (1999) Use of two-step splitt-operator approach for 2D shallow water flow computation, **Int. J. Numer. Meth. Fluids, Vol.30**, 557-575.

Rijn, L. C.van (1993) **Principles of Sediment Transport in Rivers, Estuaries and Coastal Seas,** Aqua, Amsterdam.

ASEAN Civil Engineering Conference

ISBN : 978-616-551-087-5