INTERNATIONAL JOURNAL OF SCIENTIFIC & ENGINEERING RESEARCH, VOLUME 7, ISSUE 7, JULY-2016
ISSN 2229-5518

Simulation using Computational fluid Dynamics (CFD) for Semi-Circular Breakwater during flow over it

Vivek kumar Gope, Ankit Aggarwal, Shivaraj S.M., Arkal Vittal Hegde

Abstract- Semi-Circular breakwater (SBW) is an emerging area for research on which research is still going on to find various hydrodynamic performance characteristics and flow studies over it. Due to round shape of its top it is expected that stability of SBW is high as compared to other breakwaters. This paper represents study of flow over breakwater and variation in various properties like temperature, glow properties have been studies using Computational Fluid Dynamics (CFD). Enhanced numerical modelling is done taking viscosity into account. Discretization of the geometrical model has been done such that it determines flow properties very accurately and their variation. This methodology considers improved neighbour definitions and topological consistency checks of the interface for determining a more accurate and correct interface approximation. As a result accurate results are obtained using simple options (uniform Cartesian grid, level set method). The model results are compared with free surface visualization and pressure measurements.

Index terms - Discretization; CFD; hydrodynamic; interface

---

1 INTRODUCTION

Hydrodynamic numerical simulations of flows involving multiple phases are an important research area in hydraulics. Numerical simulation is a powerful tool for understanding and predicting the complex interplay of physical and chemical processes in various media. Breakwaters are most important structure used for providing good shelters for the lee side and coastal zones. Semi-circular breakwaters are a special type of breakwaters which are circular in shape. They create calm waters and give protection for safe mooring, operating, and handling of ships, as well as protection for harbor facilities. Due to semi-circular shape of SBW models the stability against sliding for SBW is good, since the horizontal component of the wave force is smaller compared to the vertical and in addition rubble mound breakwater, the SBWs enhance the scenery. They are constructed such that their crests remain submerged in the water. With this advantage, they can not only avoid generating significant reflected wave and affecting the nearby shoreline, as other kinds of breakwaters do, but also save large quantities of engineering resources from the view of economics. Previous research has mainly focused on calculation of hydrodynamic parametes. Goda and Suzuki (1976) worked on semi-empirical formulas for SBW, Sasajima *et al.* (1994) has measured results on pressure and forces on SBW, Priya *et al.* (2000), Sundar *et al.* (2001), Sundar and Raghu (1997), have also worked on hydrodynamic performance characteristics of semi-circular breakwaters. So, simulation of flow over SBW has been done in order to study variation in various flow parameters using Computational Fluid Dynamics. CFD can be used to carry numerical simulations very efficiently. During recent decades, methods of computational fluid dynamics (CFD) were developed and implemented in modern software packages and widely used for numerical simulation of fluid flows.his paper presents study of flow properties during flow over semi-circular break waters using CFD based package FLUENT. For, doing any simulation we need a properly and appropriately meshed geometrical model which can be used as an input for carrying simulations in FLUENT. This requirement has been accomplished with the help of another software GAMBIT which has meshed our defined geometry in required manner considering proper interfacial effects between phases and the material. Proper smoothening at interface segments and nodes has been done in order to produce a good mesh.

2 PROBLEM DESCRIPTION

The breakwater model being used in the investigation involves two parts, bottom slab and semi-circular shaped caisson. Size of concrete slab to be used is 1.4m X 0.73m X 0.08m. Galvanised iron sheet has been used for fabrication purpose and radius is kept 0.6m (Figure 1). These dimensions are chosen in order to make model stable and non-overtopping.

Vivek Kumar Gope is Design Engineer at MECON Ltd. (A Govt of India Enterprises), India, PH-918969001376.
E-mail: vivekkumargope328@gmail.com
Ankit Aggarwal is PhD scholar at NTNU,
E-mail: Agarwal.ankit28@gmail.com
Arkal Vittal Hegde is Professor at National Institute of Technology Karnataka Surathkal, India. E-mail: arkalvittal@gmail.com

Sea-water waves will be hitting from sea side and it will serve as a inlet and leave from lee side and will serve as outlet. Vertical asbestos sheets are being used at distance of about 10cm from each other and kept parallel to length of flume for dissipation of generated waves and in turn, smoothening them.

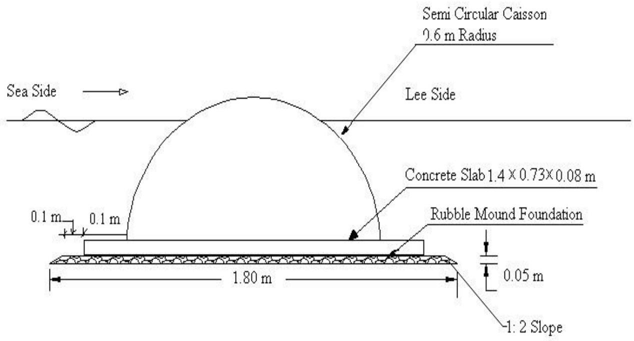

Figure 1: Basic description of problem

In this sea water flow through the semi-circular breakwater has been simulated using simulation software FLUENT. FLUENT is flow simulation software based on governing equations of CFD (Computational Fluid Dynamics) and thus can be appropriately chosen for such simulation problems accordingly. Hereby, in this setup of model, flow and flow pattern of vectors through the semi-circular breakwater has been studied and thus various vectors plots and contours have been plotted for better understanding of the results obtained.

3 GEOMETRICAL DISCRETIZATION OF THE SBW MODEL

Modelling and discretization of Semi-Circular Breakwater model has been done using software GAMBIT. Modelling process includes design of geometry, appropriate meshing and discretization of the geometry, smoothening of the nodes of the faces, projection of the nodes, building of grids, etc. This is a step by step process which has to be followed for successful modelling of the profile. Geometry which has been constructed and meshed using above mentioned step by step process in GAMBIT has been shown in Figure.2. As, this meshing is very fine after smoothening and projection of nodes at the face for the purpose of obtaining accuracy in results. Since, the only one phase, i.e. liquid phase is involved in flow, thus single phase process for modelling and simulation has to be used. While reconstructing interfaces between different materials which in turn is based on volume fraction of each constituent being present in each cell of the mesh. Many approaches have been given by many researchers for this (Kothe et. al., 1996; M. J. Shashkov, 1996; Mosso et. al., 1997; M Staley Core, 2004; Bell et. al., 1989; R. LeVeque, 1996). Also, the process of reconstruction of interface is conservative locally. Modified approach which implies the use of neighbourhood cells and dedefinitions and their checks for topological consistency of ththe interface for getting most accurate and correct apapproximately computed values and thus, giving acaccurate and refined meshing of the geometrical model.

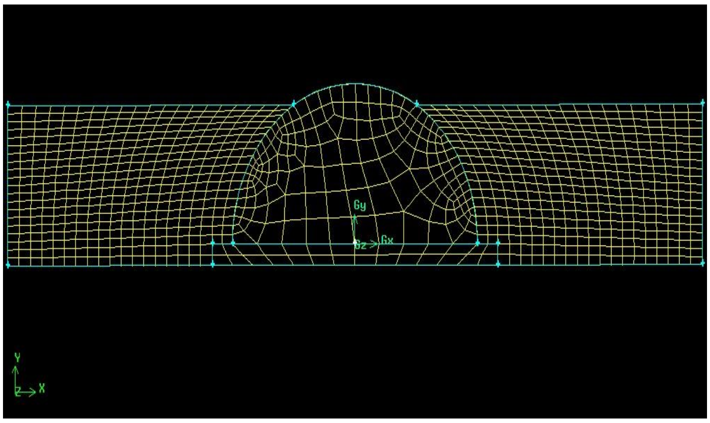

Figure 2: Meshed Geometrical model of semi-circular breakwater

Here, two dimensional approach has been used in order to reduce the complexity of the calculations in the problem. Use of two dimensional approach helps in eliminating number of nodes and cells in geometrical model as this methodology considers problem in 2-D. But, results produced by using two-dimensional approach are also accurate to same extent as that of the results produced by 3-D model. Therefore, this methodology can be used for reducing the complexity of calculations without affecting the results.

4 SIMULATION OF DISCRETIZED MODEL

For simulating the flow of water over semi-circular breakwater, the software namely FLUENT has been used. This software can tackle complex equations of CFD (Computational Fluid Dynamics) easily, which are difficult and very complex to solve manually. It makes use of pre-defined CFD equations and thus reduces time and labour to a great extent. This takes into account various basic energy equations, viscosity equations, flow equations, gravity effect etc. In the problem defined in this paper numbers of nodes being used are such that proper meshing and smoothening can be done. Types of cells being used in geometrical model are quadrilateral. Densitbased solver has been used keeping in mind the turbulence and non steadiness of fluid flow, 2-d solver, as mentioned above has used. Properties of crude oil and natural gas being used are shown in Table 1.

Table 1

Wave-specific parameters	**Experimental range**
Incident wave height, H_i (mm)	30, 60, 90, 120, 150, 180
Wave period, T (sec)	1.2, 1.4, 1.6, 1.8, 2.0, 2.2
Depth of water, d (mm)	350, 400, 450, 500
Structure-specific parameters	**Experimental range**
Radius of the semicircular caisson (mm)	600
Ratio of spacing to diameter of holes, S/D	2, 4, 6 and 8
Diameter of holes (mm)	12

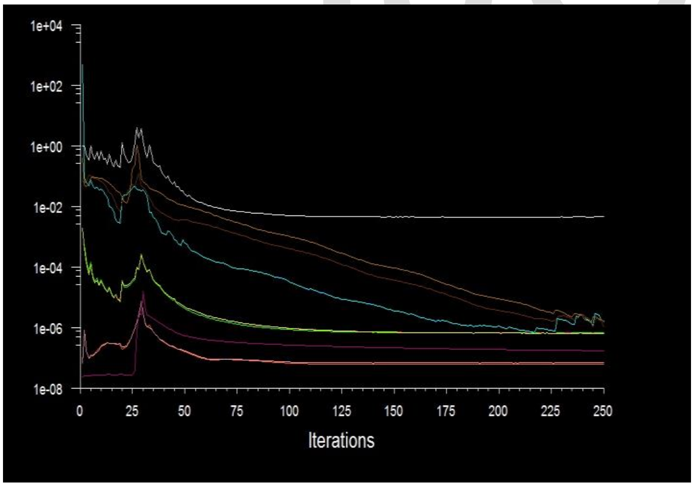

Figure.3: Iteration plots while solving for residuals

Once these values are assigned to the equations involved in FLUENT, solution is initialized for residuals and is iterated until the solution gets converged. Figure 3 shows plots for iteration until the solution gets converged has been shown for clear understanding of iteration procedure and to check that whether solution is approaching towards converging.

5 RESULTS AND DISCUSSIONS

After simulating the flow of sea water over semi-circular breakwater, study on various properties of flowing fluid has been made. Thereby, various plots, contours, and velocity vectors have been plotted in order to clearly understand the amenability of the results. Simulations have been done for 250 iterations because residual error is almost zero at this stage. Velocity vectors coloured by velocity magnitude as plotted in FLUENT shows that velocity of particles of fluid change throughout the flow (Figure. 4).

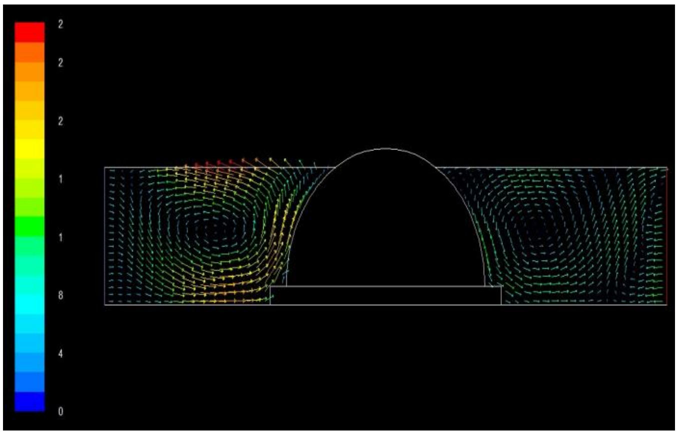

Figure.4: Velocity vectors of velocity magnitude variation

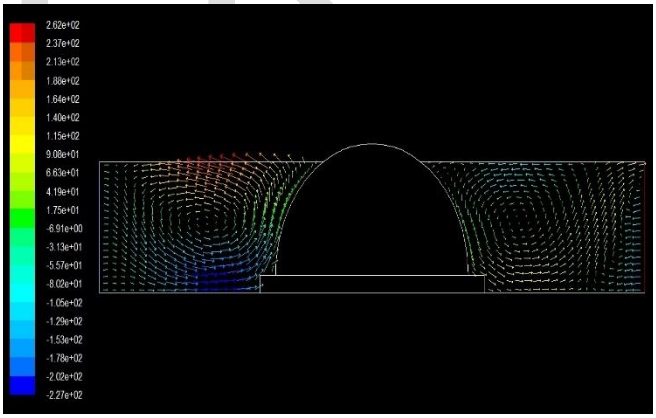

Figure.5: Velocity vectors of radial velocity variation

Direction of velocity vectors on various sections show that variation of velocity magnitude throughout the flow. Radial velocity vectors which have been shown in Figure. 5 show the variation of radial velocity throughout the flow. Velocity vectors of velocity magnitude and radial velocity clearly signifies that tangential velocity also plays a significant role in the variation of flow. So, relative tangential velocity contours have also been plotted (Figure 6). Plots of the radial velocity and velocity magnitude have been shown in Figure 7 and Figure 8, respectively.

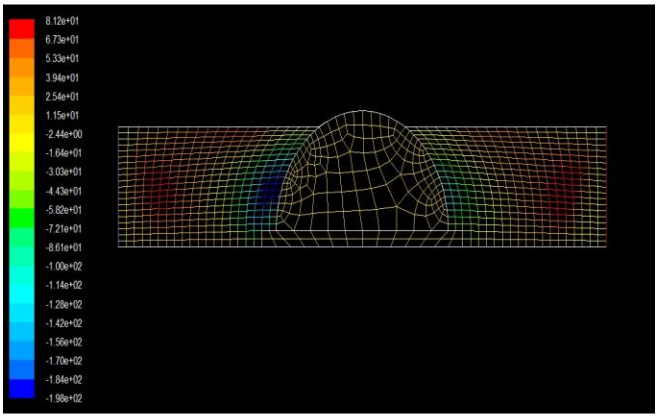

Figure 6: Relative tangential velocity contours for its variation

Contours for radial velocity have been shown in Figure 9 for clear understanding of variation of radial velocity over semi-circular breakwater. Turbulent intensity also changes at inlet part and outlet part, i.e. at sea side and lee side and it has also been shown by CFD simulation (Figure 10). Contour plots for turbulent kinetic energy are shown in Figure 11, while Figure 12 shows X-Y plots for the turbulent dissipation rate.

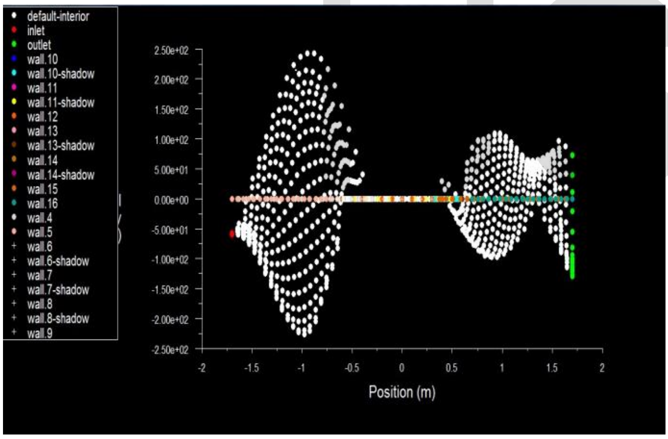

Figure 7: Plots for radial velocity variation with position of semi-circular breakwater

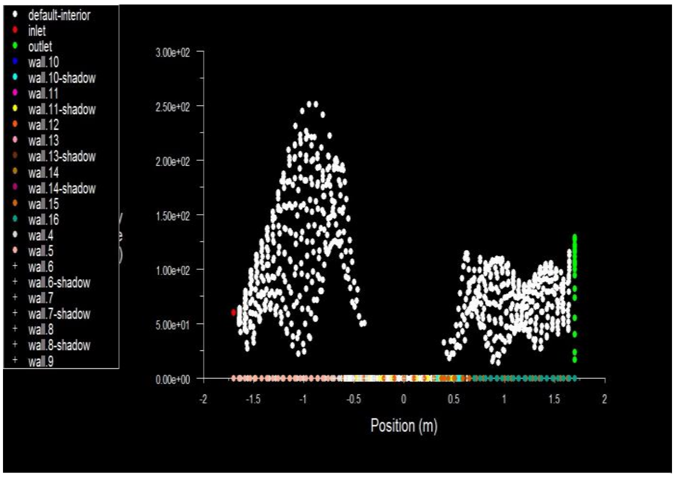

Figure 8: Plots for velocity magnitude variation with position of semi-circular breakwater

Contours for radial velocity have been shown in Figure 9 for clear understanding of variation of radial velocity over semi-circular breakwater. Turbulent intensity also changes at inlet part and outlet part, i.e. at sea side and lee side and it has also been shown by CFD simulation (Figure 10). Contour plots for turbulent kinetic energy are shown in Figure 11, while Figure 12 shows X-Y plots for the turbulent dissipation rate.

Vector plots of dynamic static pressure as shown in Figure 13, predicts that dynamic pressure also varies during simulation of flow of water in the semi-circular breakwater. For more understanding X-Y plots has also been plotted for the dynamic pressure in turn (Figure 14).

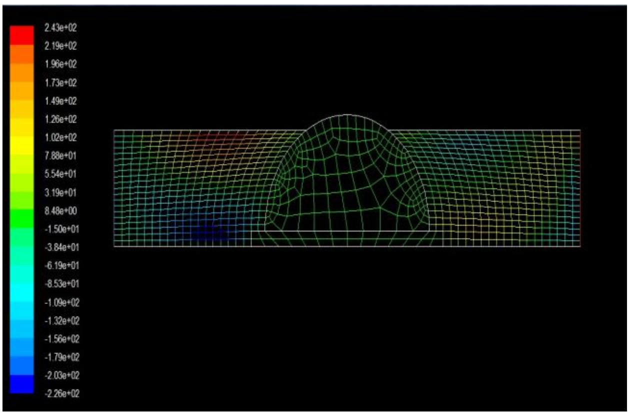

Figure 9: Contours of radial velocity variation

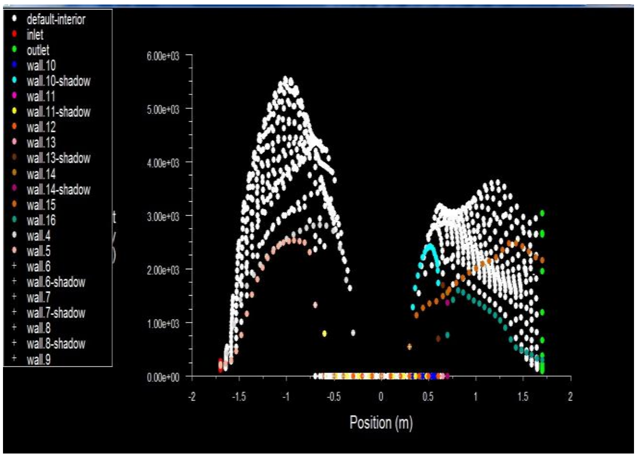

Figure 10: Plots for turbulent intensity variation with position of semi-circular breakwater

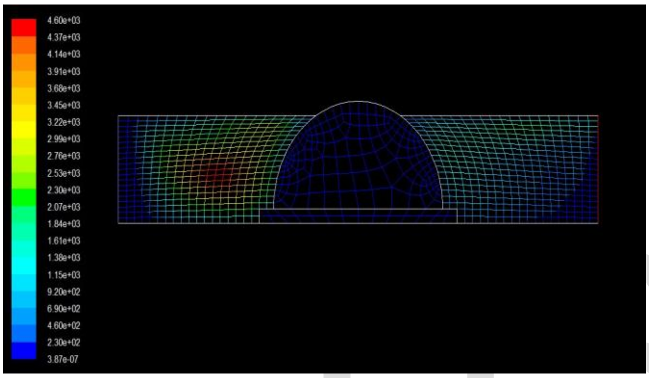

Figure 11: Contours of turbulent kinetic energy variation

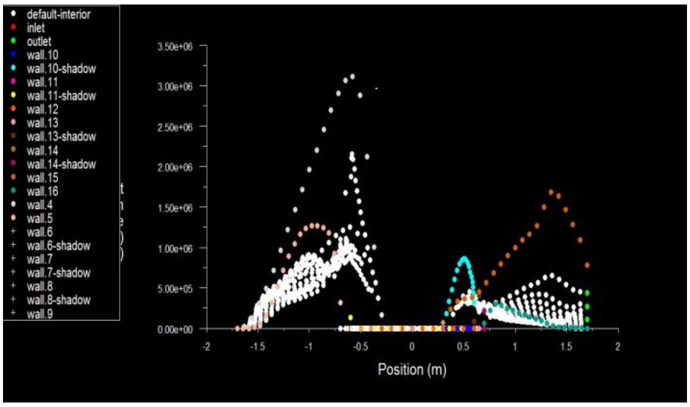

Figure 12: Plots for turbulent dissipation rate variation with position of semi-circular breakwater

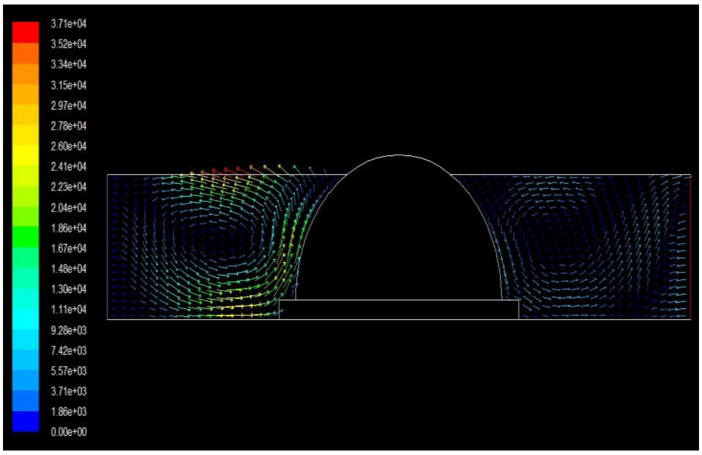

Figure 13: Vectors plots for variation of dynamic pressure

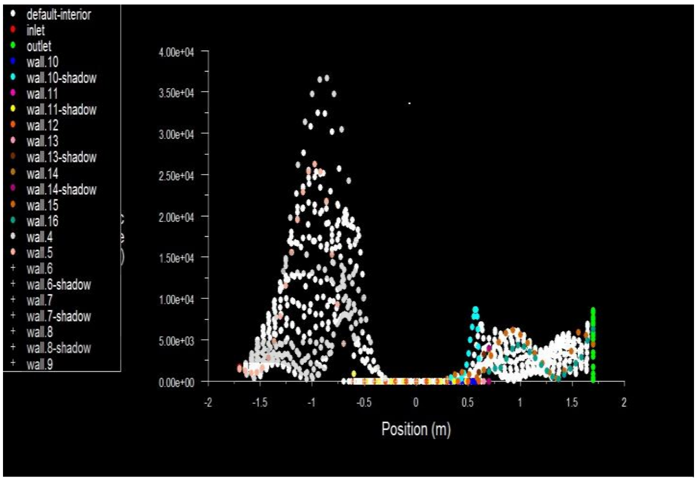

Figure 14: Plots for dynamic pressure variation with position of semi-circular breakwater

6 CONCLUSIONS

For simulation of semi-circular breakwater, it is very essential to predict the behaviour of velocity vectors, turbulence parameters and dynamic pressure in semi-circular breakwater during flow. FLUENT software can be used for this purpose with good confidence in place of expensive physical model studies. GAMBIT can be used for required geometrical modelling and meshing of geometrical model, as it gives required result with high accuracy. Direction of velocity vectors near walls and at interface also implies effectiveness of simulation approach being used in FLUENT. Hence, these geometrical models of semi-circular breakwater can be successfully used for prediction of flow, providing a fast and alternate approach for the purpose rather than complex physical model studies.

REFERENCES

[1] D. B. Kothe, W. J. Rider, S. J. Mosso, J. S. Brock, and J. I. Hochstein(1996) Volume tracking of interfaces having surface tension in two and three dimensions. In Proceedings of the 34th Science Meeting, Reno, NV, AIAA.
[2] Goda, Y. and Y. Suzuki (1976) Estimation of incident and reflected waves in random wave experiments. Proceedings of the 15th International Conference Coastal Engineering, (ICCE 76), ASCE, Honolulu, Hawaii, pp: 828-845.
[3] J. B. Bell, P. Colella, and H. M. Glaz (1989) A second-order projection method of the incompressible Navier-Stokes equations. Journal of Computational Physics, v.85, pp: 257–283.
[4] Kochevsky, A. N., Nenya, V. G., (2004) Contemporary approach for simulation and computation of fluid flows in centrifugal hydromachines.
[5] Kochevsky, A. N., (2004) Possibilities for simulation of fluid flows using the modern CFD Software tools.
[6] M. J. Shashkov, (1996) Conservative Finite-Difference Methods on General Grids. CRC Press, Boca Raton, FL.
[7] M Staley Core, (2004) Conservative remapper, Technical Report LAUR-04-8104, Los Alamos National Laboratory.
[8] Priya, M.S.K., K.A. Roopsekhar, V.Sundar, R. Sundaravadivelu, K.U.Graw and S.Knapp (2000). Hydrodynamic pressures on semicircular breakwaters. CD Proceedings of the 4th International Conference on Hydroscience and Engineering, Sept. 26-29, IAHR, Seoul, South Korea, pp: 1-10.
[9] R. LeVeque, (1996) High-resolution conservative algorithms for advection in incompressible flow. SIAM Journal of Numerical Analysis, v.33, pp: 627–665.
[10] Sasajima, H., T. Koizuka and H. Sasaayama, (1994). Field demonstration test of a semicircular breakwater. Proceedings of HYDROPORT'94, International Conference on Hydro-Technical Engineering for port and Harbour Construction, Oct. 19-21, Port and Harbour Research Institute , Yokosuka, Japan, pp: 593-610.
[11] S. J. Mosso, B. K. Swartz, D. B. Kothe, and R. C. Ferrell, (1997) A parallel, volume-tracking algorithm for unstructured meshes. In P. Schiano, A. Ecer, J. Periaux, and N. Satofuka, editors, Parallel Computational Fluid Dynamics: Algorithms and Results Using Advanced Computers, p. 368–375. Elsevier Science B.V.
[12] Sundar, V. and D. Raghu, (1997). Wave induced pressures on semi-circular breakwater. Proceedings of the 2nd Indian national Conference on harbour and Ocean Engineering, (INCHOE'97), Cochin University of Science and Technology, Thiruvanathpuram, India, pp: 278-287.
[13] Sundar, V., R. Sundaravadivelu, G. Dhinakaran, K.A. Roopsekhar and K.U. Graw, (2001). Pressures and forces on semicircular breakwaters due to regular waves. Oceanic Eng. Int. 5, pp: 73-83.