18th Australasian Fluid Mechanics Conference
Launceston, Australia
3-7 December 2012

The prediction of wave patterns at large distances from a moving body in a confined channel

Mohammadreza Javanmardi, Jonathan R. Binns, Martin R. Renilson and Giles Thomas
Australian Maritime College, University of Tasmania, Launceston, Tasmania 7250, Australia

Abstract

In this study we have investigated the ability of computational fluid dynamics (CFD) to predict far field wave propagation parameters generated by vessels at forward speeds in a confined channel.

A model pressure source has been tested in the Australian Maritime College Towing Tank with wave probes located at a range of lateral distances through the channel which measured the parameters of the generated waves.

The CFD software ANSYS Fluent was used to simulate the same conditions using the finite volume method. The numerical results are compared with experimental data for different lateral distances from the pressure source. The comparisons show that the numerical approach is able to satisfactorily predict the wave parameters in the far field, as well as the near field.

Introduction

Vessels operating at speed in open or restricted waters generate waves and prediction of the characteristics of these waves during the design phase can be critical. Such waves can cause many problems for both other vessels and the environment[1]. Some of the most important effects and problems are:

- causing extreme water level drawdown and return currents in confined channels;
- generating surge waves in shallow waters;
- disturbing other vessels in marinas, ports and harbours;
- causing shoreline erosion and riverbank instability;
- impacting on marine life in coastal wetlands.

As a result, there is a need to develop tools for predicting the ship generated near-field waves and their propagation to the far field. Due to the influence of wave effects on ships and the environment, significant research has been conducted to date. The majority of investigations have been conducted by theoretical [2, 3] or experimental approaches [4, 5]. Most numerical investigations have focused on drag prediction and vessel dynamics. Typically, the finite volume method has been used with significant far-field damping for floating structures, reducing significantly the accuracy of predicted wave heights [6]. In this study we have investigated the ability of computational fluid dynamics to predict wave propagation parameters in the far field as generated by vessel forward speed in a confined channel.

A pressure source model was tested in the Australian Maritime College towing tank at different speeds. Wave probes at different lateral distances captured the generated wave parameters. Next, simulations were conducted using ANSYS Fluent software in the same condition as the experiment and finally, through the comparison of computed and measured results, applicability of the numerical method is examined.

Experimental tests

A wavedozer model was used as a pressure source during the experiments. The wavedozer model [7] is a wedge shape model with a constant beam (figure 1). The main particulars of the wavedozer are listed in table 1.

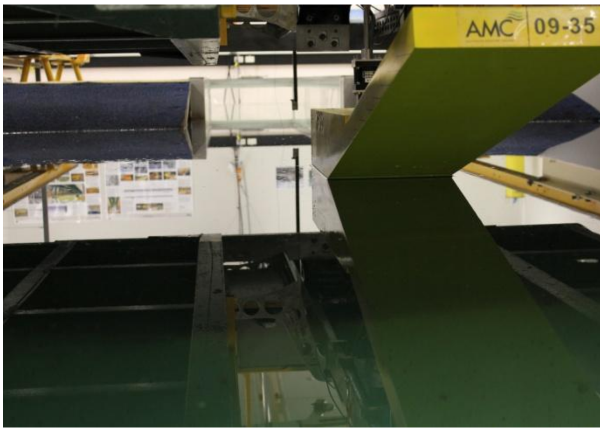

Figure 1: Wavedozer attached to the carriage in the AMC Towing Tank prior to a test run

Table 1: Wavedozer Principal Particulars

Length (m)	1.5
Beam (m)	0.3
Draft (m)	0.1
Angle of attack (deg)	14

Where the angle of attack is the relative angle between the entry surface and the water surface. This model was tested in the Australian Maritime College towing tank, which is 100 m in length and 3.5 m in width. The water depth was 1.5 m in all of the tests. The model was attached to the carriage using a two post towing system, with the model fixed and therefore no sinkage or trim was permitted during a test run. Three resistance wave probes were positioned at 0.75, 1.0 and 1.25 m from the centre-line of the model to record the elevation of the vessel-generated waves with respect to time (Figure 2). A load cell was installed at the connection between each towing post and the model to measure the vertical and longitudinal horizontal forces. The model was tested for varying Froude depth numbers from 0.26 to 0.99.

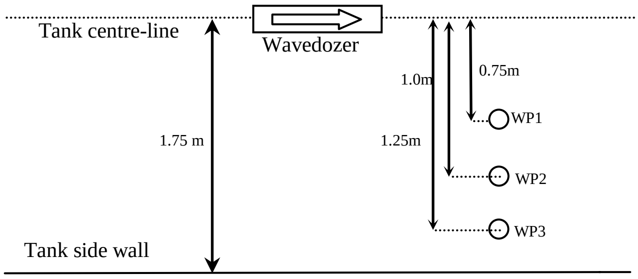

Figure 2: Layout of probes relative to wavedozer pressure source

Numerical Simulations

The CFD software FLUENT version 12.1 was used as the flow solver in this study. The governing equations are three-dimensional Reynolds averaged Navier-Stokes equations for incompressible flows. The volume of fluid approach was used with a time-dependent and explicit time discretization scheme employed to solve the equations. The SIMPLE algorithm was used for the pressure-velocity coupling and the PRESTO scheme for the pressure interpolation. The k-epsilon model with the standard wall function was utilised for turbulence modelling. The 2nd order upwind scheme was used for solving the momentum equations and the High Resolution Interface Capturing scheme (HRIC) for the solution of the volume fraction equations.

Figure 3 shows the computational grid domain which consisted of 6 m upstream of the model and 13.5 m downstream of the model. As with the experiments, the heave and trim of the model were fixed. As the flow has a plane of symmetry about the centre plane, to reduce the processing time, half of the domain was used; the origin of the coordinate system was located at the middle of the model. The open channel boundary condition was used to specify the inlet and outlet boundary condition. A symmetry plane was used along the centre plane, and the remaining boundary surfaces along the exterior of the domain were set to no-slip wall conditions. The mesh size was chosen based on the grid independence study previously completed by the authors [8]. According to that study, by using eight times more cells, the wave height prediction would improve by less than 2%. Therefore this size of cell was chosen as the optimal.

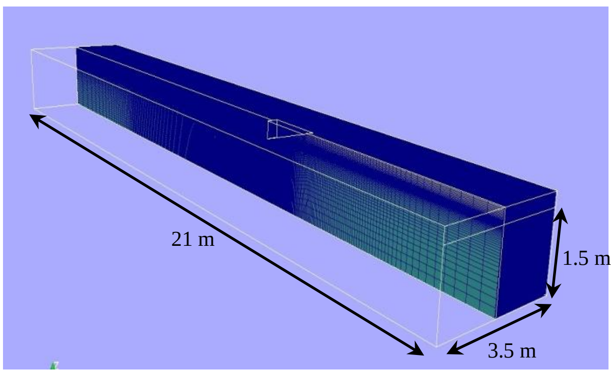

Figure 3: Computational grid domain

Results and Comparisons

The results of the numerical simulation have been compared with experimental data in Figures 4 to 11. Figure 4 shows the horizontal force (drag) results for both the experimental results and the numerical solutions, whilst Figure 5 presents the vertical force for different speeds. In Figure 4 and 5, the error bars show the experimental uncertainty which was estimated to be 5%. It is clear that the simulation results are in excellent agreement with the experimental data with respect to these horizontal and vertical forces. For the full range of Froude depth number investigated the numerical results fall within the experimental error bars for each speed tested.

Sample time histories of the wave elevation for Froude depth numbers 0.75 and 0.99 for the three wave probes are presented in Figures 6 and 11.

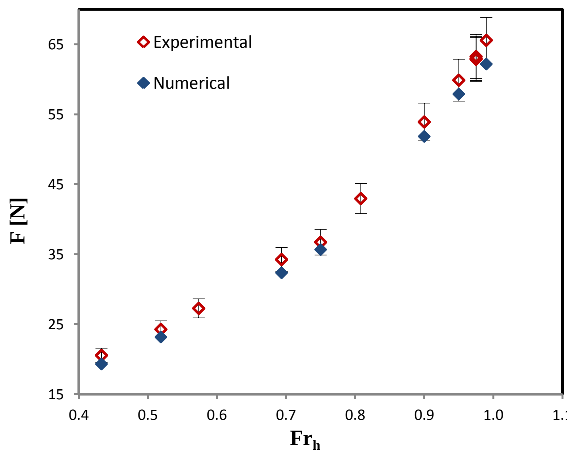

Figure 4: Comparison of experimental and numerical wavedozer horizontal force (drag) for varying Froude depth number.

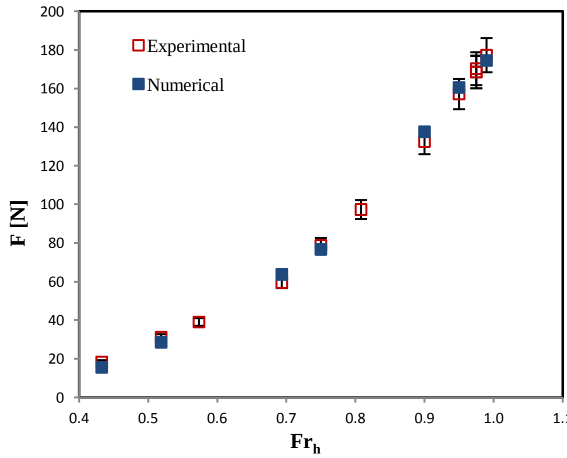

Figure 5: Comparison of experimental and numerical wavedozer vertical force for varying Froude depth number.

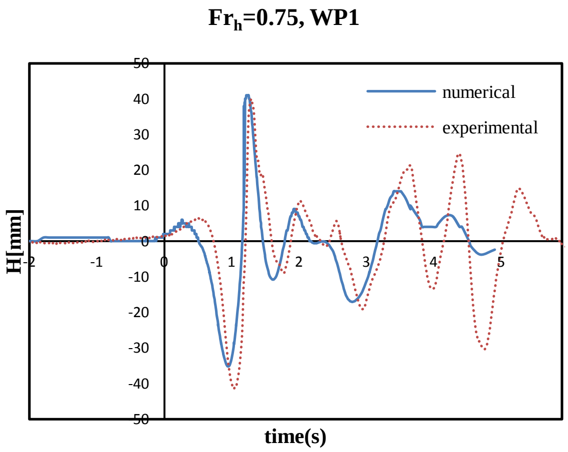

Figure 6: Wave time history for Fr$_h$=0.75 in 0.75m lateral distance

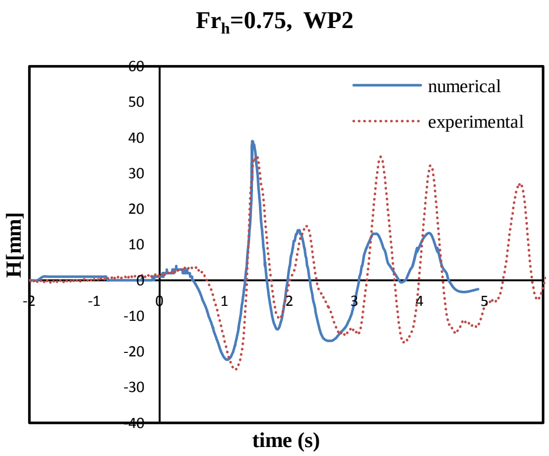

Figure 7: Wave time history for Fr$_h$=0.75 in 1.0m lateral distance

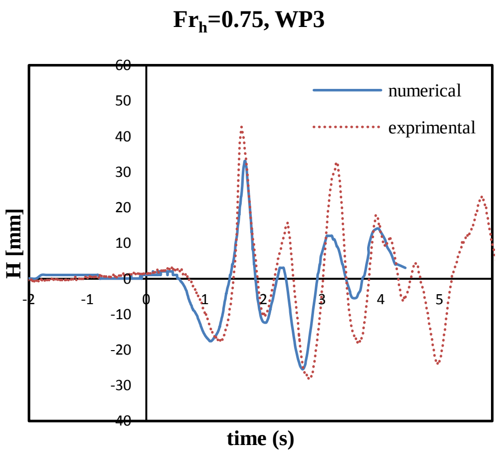

Figure 8: Wave time history for Fr$_h$=0.75 in 1.25m lateral distance

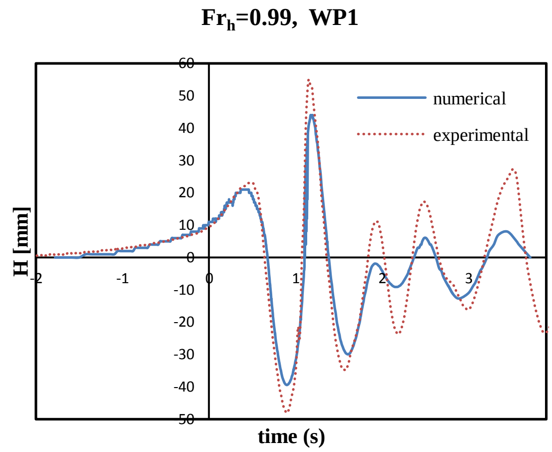

Figure 9: Wave time history for Fr$_h$=0.99 in 0.75m lateral distance

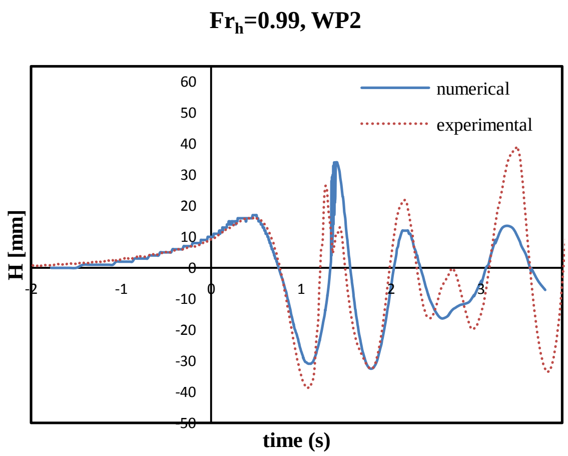

Figure 10: Wave time history for Fr$_h$=0.99 in 1.0m lateral distance

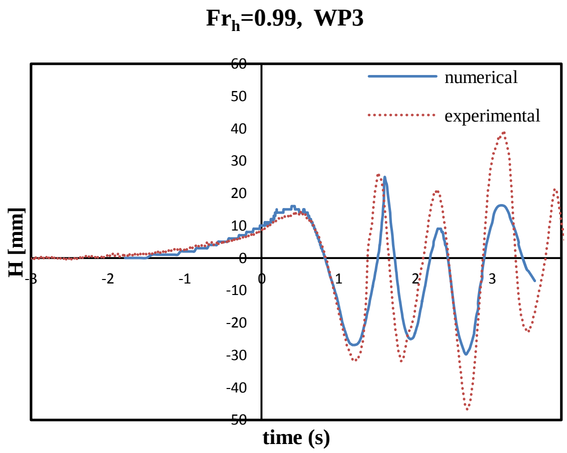

Figure 11: Wave time history for Fr$_h$=0.99 in 1.25m lateral distance

The wave elevation time histories shown in Figures 6 to 11, demonstrate that the CFD simulation is able to accurately predict the wave patterns at different lateral distances, particularly the first two wave cycles in the system. The correlation for the subsequent waves is less satisfactory and this may be due to the effect of the flow boundary conditions, high aspect ratio cells in the downstream part of computational domain and resulting numerical diffusion.

Figures 12 and 13 show the experimental and the numerical free surface elevations. The qualitative comparison between the experimental and the numerical free surface elevation is quite good.

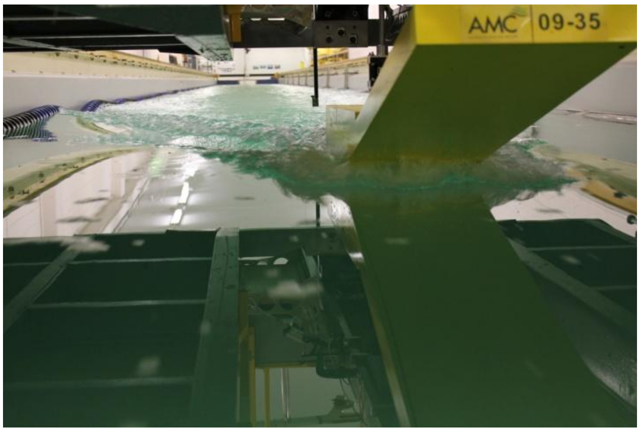

Figure 12: snapshot of free surface elevation around the wavedozer

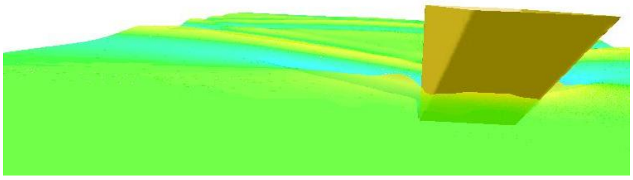

Figure 13: computed free surface elevation

Conclusion

The results of an initial investigation of moving body waves in a confined waterway have been presented. The experimental investigations were conducted in the Australian Maritime College towing tank and the commercial software CFD software Fluent with VOF technique was used to simulate the generated wave around the moving body in restricted waterway.

The comparison of the numerical predictions with the experimental data has confirmed that the numerical approach can be used to accurately predict the vertical force, drag and wave elevations generated by this wavedozer in a confined waterway for the area of greatest interest to the project. However, it was found that well downstream of the model, the predicted wave pattern begins to vary considerably for the numerical predictions. It has been proposed that this is due to the proximity of numerical boundary conditions and numerical diffusion resulting from increases in cell aspect ratio in this region.

The presented results indicate that the Fluent CFD software is a viable tool in an extreme flow condition and should be considered for further investigations.

Acknowledgements

This research has been conducted as part of a collaboration research project between Webber Wave Pools, TU Delft, and the Australian Maritime College at the University of Tasmania with funding from the Australian Research Council, project ID: LP0990307.

References

1. Varyani, K.S., *Full scale study of the wash of high speed craft.* Ocean Engineering, 2006. **33**(5-6): p. 18.
2. S. Chandraprabha, A.F.M., *A numerical preiction of wash wave and wave resistance of high speed displacement ships in deep and shallow water*, in *Faculty of Mechanical Engineering Network of Thailand 18.* 2004: Khon Kaen, Thailand.
3. M. R. Varnousfaaderani, M.J.K., *Numerical simulation of waves generated by ships in shallow water.* journal of ship technology, 2008. **4**(2).
4. E. Fontaine, M.P.T., *On the prediction of Nonlinear free-surface flows past slender hulls using 2D+t Theory: the evolution of an idea*, in *Fluid dynamics of vehicles operating near or in the air-sea interface*. 1998: Amsterdam, The Netherland.
5. R. Henn, S.D.S., *Influence of canal topography on ship waves in shallow water*, in *16th International workshop on water waves and floating bodies*. 2001: AKI grand Hotel, Hiroshima.
6. Y. Kim, *Artificial Damping In Water Wave Problems II: Application to Wave Absorption.* The International Society of Offshore and Polar Engineers, 2003. **13**(2): p. 5.
7. A. Driscoll, M.R.R., *The wavedozer. A system of generating stationary waves in a circulating water channel*. 1980.
8. M. Javanmardi, J.B., M.R. Renilson, G. Thomas, S. Schmied, R. Huijsmans, *The formation of surfable waves in a circular wave pool-comparison of numerical and experimental approaches*, in *31th International Conference on Ocean, Offshore and Arctic Engineering*. 2012: Rio de Janeiro, Brazil.