Numerical Investigation on Flow Making System in Multi-purpose Basin

Zheng Xu* and Xiaohui Su

School of Hydraulic Engineering, Dalian University of Technology, Dalian, China, 116023

*Xuzheng123@mail.dlut.edu.cn

Abstract. In this paper, numerical investigation on flow field of flow making system is carried out in the 1:10 scaled model of the multi-purpose basin for the deep sea engineering research center of Dalian University of Technology. According to the requirement of obtaining specified flow velocity in the working area, two kinds of numerical methods simulating pump are used. The calculation results show that the pump simulating method with controlling boundary conditions at inlet and outlet is not valid in the presence of free water surface. The problem is solved by the second pump simulating method in which the pump rotation is simulated in the multiple frame of reference (MRF) system. It is believed that the establishment of current model will provide an effective calculation tool for the simulation of flow characteristics of the deep water flow system.

1. Introduction

In order to upgrade the offshore engineering test technology [1,2], Dalian University of Technology (DUT) is planning to build an oceanic deep water multi-purpose basin with functions of flow making, waves making and wind making. The flow making system is the most important part of the project [3-6].

The pool of flow making system in multi-purpose basin is designed as 50m long, 30m wide and 12.9m deep. The designed free water surface height is 12m. Based on the initial design, 14 water pumps are equipped. The depth of the floating bottom can move in the range of 0~10m in the water depth direction. In order to make the flow of water into the multi-purpose basin and achieve a stable flow, a few of perforated walls and deflectors are set in the system. A working area with $10\times10m^2$ of the middle basin is set for testing.

In this paper, the 1:10 scaled model of the making flow system is numerically investigated. Since those 14 pumps and channels of inflow system are identical, only one of the inflow channels is numerically simulated.

2. Numerical Methods

Reynolds-averaged multiphase incompressible fluid Navier-Stokes equations and continuity equations are implemented to describe flow motion in the flow domain. VOF (Volume of Fluid) method is used for free surface capturing [7,8] .

The standard k-ε model is used for dealing with turbulence happening in the flow field. The standard k-ε model is mainly based on the turbulent kinetic energy and turbulent dissipation rate of two parameters to describe the turbulence.

When studying the calculation domain of the pump impeller, the multiple frame of reference (MRF) is adopted. The pump and the main body of flow making system belong to rotating part and stationary part separately.

3. Pump Simulation I

3.1 Model setup

3.1.1 Geometry description. A single inflow channel with 5m length, 0.215m width, 1.29m height and free water surface height 1.2m is modelled. The height of floating bottom is set to 0.8m. Flow making system model diagram is shown in Fig. 1. The perforated wall model is shown in Fig. 2.

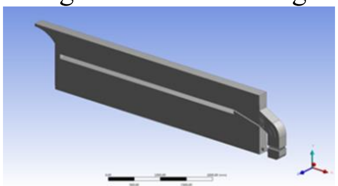


Figure 1. Geometry of flow making system model.

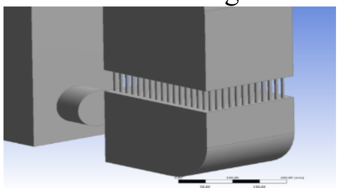


Figure 2. Zoomed in Perforated wall.

3.1.2 Mesh information. The number of grid cells in the entire model is about 890,000 after the mesh convergent test. The overall grid diagram of flow making system and the grid of perforated wall are shown in Fig. 3 and Fig. 4, respectively.

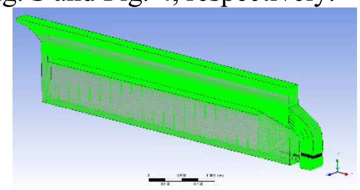


Figure 3. Grid of flow making system.

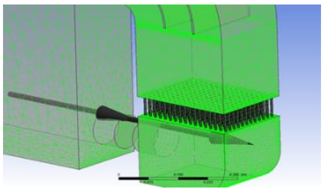


Figure 4. Grid of perforated wall.

3.2 Boundary conditions and calculation conditions

Entrance boundary is set as total pressure (11800pa). Export boundary is set as mass flow rate (2.752kg / s). Wall condition is set as no slip wall. The two side walls of the main body of the flow making system are set as the symmetrical boundary. Water and air interface is set as free boundary. The boundary conditions are denoted in Fig. 5.

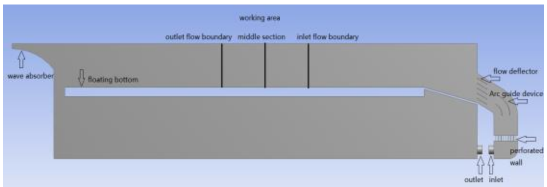


Figure 5. Boundary conditions of the flow making system.

3.3 Calculation results and analysis

3.3.1 Streamlines of flow making system and free water surface. The position of free water surface captured by the VOF method at 60s is plotted as green plane in Fig. 6. The height of free water surface at that time is about 1.25m. Turbulent flow of water entering into making flow system through inlet channel is obvious, and there is a large whirlpool.

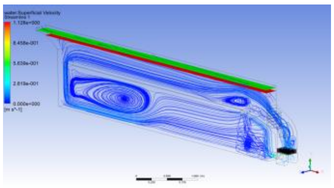


Figure 6. Free water surface and streamlines at 60s.

3.3.2 Vertical velocity profiles in working area boundaries. In this paper, focus is made on flow field characteristics in the working area. The sketch of working area and investigating lines are plotted in Fig. 7. It states that the inlet flow boundary (X = 3.0m) is set at the right side of the working area, the middle section (X = 2.5m) is set in the center of working area, and the outlet flow boundary is set at the left side of the working area (X = 2.0 m).

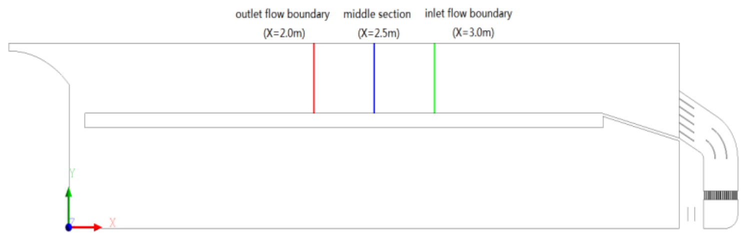


Figure 7. Boundaries of working area.

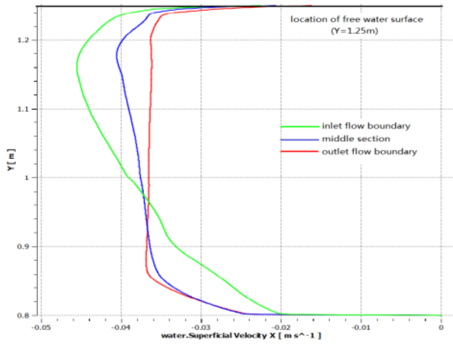


Figure 8. Vertical velocity profiles.

In Fig. 8, the green line represents the inlet flow boundary, the blue line represents the middle section, and the red line represents the outlet flow boundary, respectively. The average velocity at the inlet flow boundary is larger than those values at the middle section and the outlet flow boundary.

The instantaneous results at 60s were chosen to check mass convergence. It found that the mass flow at inlet is 3.28 kg /s while the value at outlet is 2.70 kg/s, indicating that this simplified pump simulation increases the non-conservative amount of water mass in the system.

4. Pump Simulation II

4.1 Model setup

4.1.1 Geometry description. The model is 5m long, 0.215m wide, 1.29m high, the height of the free water surface is still 1.2m and the height of the floating bottom is set to 0.63m. A four-blade water pump model is inserted into the rotation domain. Model diagram is shown in Fig. 9, the pump model is shown in Fig. 10.

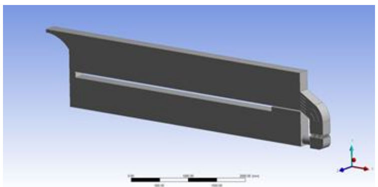


Figure 9. Geometry of flow making system.

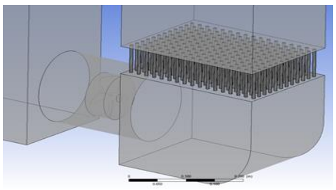


Figure 10. Water pump model.

4.1.2 Mesh information. The amount of total grid is about 5,370,000, in which the amount of rotating domain mesh reaches 3,620,000, the amount of stationary domain mesh reaches 1,750,000. Schematic diagram of the overall model of the flow making system is shown in Fig. 11, the pump model diagram is shown in Fig. 12.

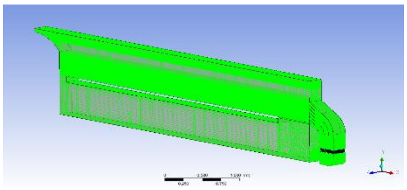


Figure 11. Overall grid of flow making system.

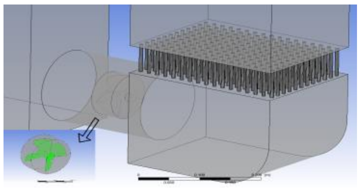


Figure 12. Grid of water pump.

4.2 Boundary conditions and calculation conditions

The calculated boundary includes the wall (no slip), the symmetrical boundary and the interface between the rotating area of the pump and the stationary area of flow making system, the sketch of boundaries is plot in Fig. 13.

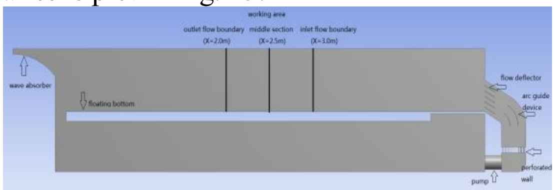


Figure 13. Boundary conditions of flow making system.

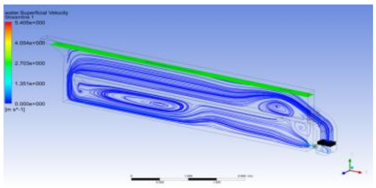


Figure 14. Streamlines in flow field at 55s.

In order to obtain specified velocity, 0.032m/s, in the working area in which the corresponding flow velocity of prototype is 0.1m/s according to the gravity similarity criterion, a few of rotating speeds of pump, range from 800rpm to 1600rpm with 200rpm interval, have been tried. Finally, rotation rate, 1200rpm, is found to be proper one.

4.3 Calculation results and analysis

4.3.1 Streamlines of overall flow field. There exists a vortex in front of working area, which is believed due to flow deflector. After the vortex, flow passes through channel smoothly and there is no vortex any more before flow reaches the end of the channel. Streamlines in overall flow field at 55s is shown in Fig. 14.

4.3.2 Vertical velocity profiles in working area. Same as the treatment of vertical velocity profile investigation in Section 3, velocity profiles in the working area are obtained through the velocity in the vertical direction in the working area set in Fig.15.

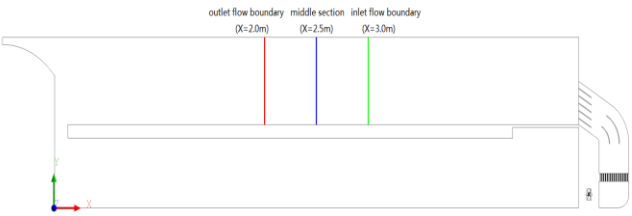


Figure 15. The sketch of working area.

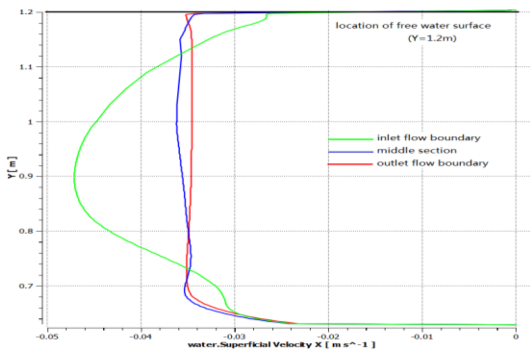


Figure 16. Velocity profiles in the working area boundary at 55s.

Fig.16.shows vertical velocity profiles at different positions in the working area according to the results at 55s. The profiles show that velocity varies greatly along vertical direction and the averaged value is larger at inlet flow boundary (green line), compared with the other two colorful lines. It can be explained that the energy of water flow is affected by the vortex generated at the position of front of working area. At the middle section (blue line) of the working area, the variation of water flow

velocity in vertical direction decreases. The velocity at the outlet flow boundary of the working area (red line) tends to be uniform in the vertical direction. Therefore, it is reasonable to carry out model tests near the outlet flow boundary in the working area.

4.3.3 Velocity histories in specified points. There are totally nine points specified here for the output of velocity histories. Point 1 (Y = 0.7m), point 2 (Y = 0.9m) and point 3 (Y = 1.1m) are located at the inlet flow boundary of working area (X = 3.0m). Point 4 (Y = 0.7m), point 5 (Y = 0.9m) and point 6 (Y = 1.1m) are located at the middle of the working area (X = 2.5m). Point 7 (Y = 0.7m), point 8 (Y = 0.9m) and point 9 (Y = 1.1m) are at the outlet flow boundary of the working area (X = 2.0m). Fig. 17 shows location of the points. Fig. 18 show velocity histories of points 1-3, points 4-6 and points 7-9, respectively.

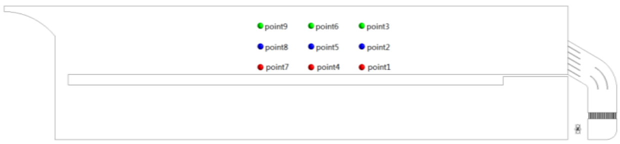
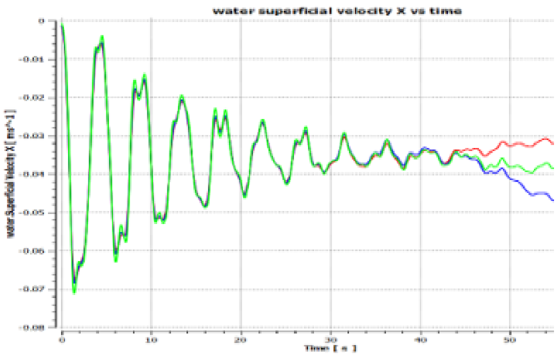
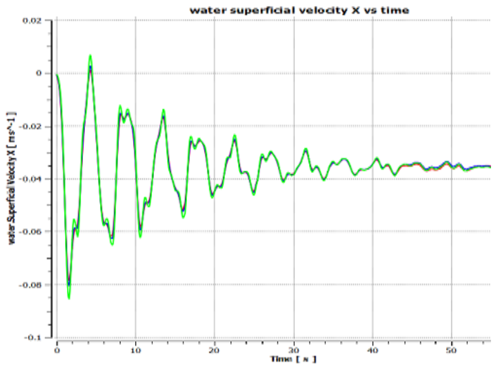


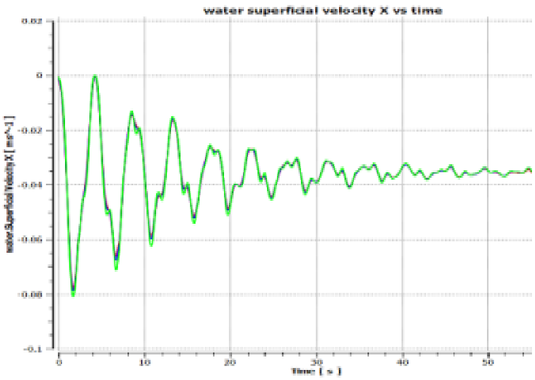
Figure 17. The layout of specified points.



(a)Points 1-3



(b) Ponts 4-6



(c) Points 7-9

Figure 18. Water velocity histories of points.

Fig. 18 shows that flow field in working area becomes stable during 45 seconds running. The reason that vertical velocities at the inlet flow boundary begin different is mainly due to the effect of wall boundary of floating bottom. The profiles at of points 7-9 at the outlet flow boundary matches very well.

4.3.4 Pump hydraulic performance calculation. According to the calculation results in the case of rotating speed 1200rpm, those hydraulic characteristic parameters of the calculated pump are listed in Table 1.

Table 1. Pump hydraulic characteristics

Rotation speed N（rpm）	Inlet total pressure（Pa）	Outlet total pressure（Pa）	Mass flow rate（kg/s）	Torque on blade M（N.m）	lift（m）	Shaft power（KW）	effective power（KW）	efficiency（%）
1200	10426.24	13524.45	4.28	0.39	0.32	0.049	0.0134	27.3

5. Conclusions

In this paper, numerical investigation on flow field of flow making system is carried out in the 1:10 scaled model of the multi-purpose basin for the deep sea engineering research center of Dalian University of Technology. A few of conclusions are made as follows:

(1) The pump simulating method with controlling boundary conditions at inlet and outlet is not valid in the presence of free water surface. Non-conservation flow rate is produced. The problem could be solved by the second pump simulating method in which the pump rotation is simulated in the multiple frame of reference (MRF) system.

(2) According to the results of the MRF method, the velocities in the vertical direction are almost uniform at the outlet boundary in the working area. Therefore, it is suggested to conduct experiments near the outflow boundary of the working area.

(3) The effective power of this pump is predicted about 0.0134kW and the efficiency is about 27.3% at 1200rpm. The averaged flow velocity in working area is about 0.035m/s, which is slightly larger than the ideal flow rate of 0.032m/s.

Acknowledgement

This research is supported by the Liaoning Natural Science Foundation, project number 2015020629. The financial support is gratefully acknowledged.

References

[1] Yucheng Li 1998 The New Development of Offshore Engineering Technology. *J. China Offshore Platform,* **13**, pp9-12

[2] Runpei Li, Yonghe Xie and Zhi Shu 2003 A Review on the Technical Development of Deep Water Offshore Platform. *J. China Ocean Platform,* **18**, pp 1-5.

[3] Chenghong Mao, Jianmin Yang, Tao Peng and Xuefeng Wang 2007 The numerical simulation and analysis of water commutating equipment in current generation system of deepwater offshore basin. *Journal Of Hydrodynamics,* **22**, pp 483-490.

[4] Fen Li and Zaojian Zou 2003 Research Condition and Development Tendency of the Floating Structures. *Journal of Wuhan University of Technology (Transportation Science & Engineering).* **27**, pp 682-686.

[5] Guoyu Gao, Yongchun Xu and Haibei Lin 2015 Design of water flow engineering system for a deepwater offshore basin. *J. Water & Wastewater Engineering.* **51**, pp 56-58.

[6] Haining Lv, Jianmin Yang and Tao Peng 2006 Research on parameters of perforated wall in current generation system of deepwater offshore basin. *Journal of Hydrodynamics,* **21**, pp 198-204.

[7] Bin Liao and Shanqun Chen 2011 Simulation of water Oscillation Flow Field by VOF Method *J. Journal of Yangtze River Scientific Research Institute,* **28**, pp 27-30.

[8] Ling Li, Yongcan Chen and Yonghong Li 2007 Three-dimensional VOF model and its application to the water flow calculation in the spillway. *Journal of Hydroelectric Engineering* **26**, pp 83-87.