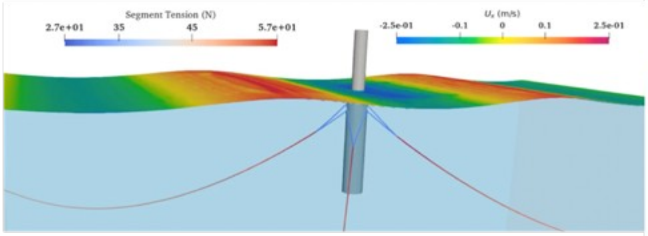
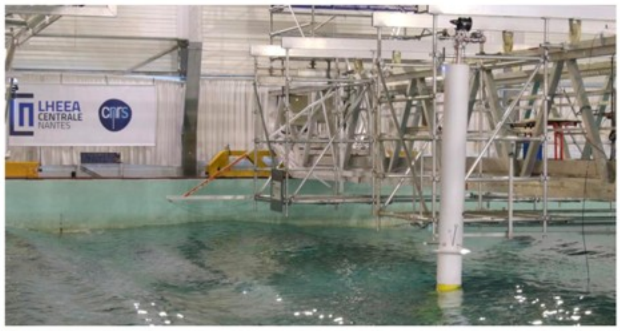
19e Journées de l'Hydrodynamique
26 – 28 novembre 2024, Nantes





SIMULATION CFD DE LA MANŒUVRABILITÉ EN EAU PEU PROFONDE ET CONFINÉE

CFD MANOEUVRING SIMULATION IN SHALLOW AND CONFINED WATER

G. DENG[1], K. ELOOT[2], W. VAN HOYDONCK[2]
ganbo.deng@ec-nantes.fr

[1] LHEEA, Ecole Centrale de Nantes, Nantes, France
[2] Flanders Hydraulics, Antwerp, Belgique

Résumé

Ce papier est consacré à la simulation CFD de la manœuvrabilité de bateau en eau peu profonde en milieu confiné. La prédiction numérique de cercle de braquage du côté bâbord avec l'angle de gouvernail à 35° ainsi que les manœuvres zigzag -20°/5° et 20°/-5° sont comparés aux résultats d'essai obtenus par Flanders Hydraulics (FH) et par MARIN pour le tanker KVLCC2 à l'échelle 1/75 avec un nombre de Froude Fr=0.0647 et un ratio de profondeur sur tirant d'eau h/T=1.2 aussi bien en milieu non-confiné comme en milieu confiné.

Summary

This paper is devoted to CFD simulation of ship manoeuvring in shallow and confined water. Numerical predictions for the port side turning circle manoeuvre with 35° rudder angle and the zigzag -20°/5° and 20°/-5° manoeuvres are compared with experimental data obtained by Flanders Hydraulics (FH) and by MARIN for the KVLCC2 ship model at scale 1/75 with Fr=0.0647 and depth to draft ratio h/T=1.2 both for non-confined and confined configurations.

I – Introduction

With the advance of computational methods and the increase in computational power, CFD simulation is becoming a useful tool for ship manoeuvring prediction. In the series of international workshop dedicated to the assessment of the predictive capability of different approaches for ship manoeuvring (SIMMAN 2008, 2014, and 2020), CFD submissions keep increasing. More and more results of CFD simulations for ship manoeuvring have been published (Wang[1], Aram[2], Deng[3]). However, due to specific numerical difficulty for ship manoeuvring simulation in shallow water, few published results are available for shallow water, especially for confined shallow water. Carrica et al. [4] have computed a zigzag manoeuvre with actual propeller approach without taking into account bank layout. They reported that the rudder rate has an important impact on the prediction of the overshoot angle in the zigzag manoeuvre. Kim et al. [5] have performed a simulation for the same zigzag manoeuvre with the same conditions except for the scale factor. They have also performed turning circle simulations with several water depth to draft ratios but without validation against measurement data. All simulations have been performed in non-confined configuration. During the last SIMMAN 2020 workshop held in 2023, there were 29 CFD submissions for deep-water ship manoeuvring predictions, while for the shallow water test cases, there were only two submissions. The lack of CFD validation work for shallow water manoeuvring applications, especially in confined water motivate us to perform CFD simulation for several turning circle and zigzag manoeuvres. Results for the KCS test case have been published in a previous paper Deng et al. [6]. The present paper is devoted to the same tasks with focus on the KVLCC2 test case.

II – Numerical approach

Numerical simulations are performed with the ISIS-CFD flow solver, available as a part of the FINE^{TM}/Marine computing suite distributed by Cadence Design Systems. It is an incompressible unsteady Reynolds-averaged Navier-Stokes (URANS) solver mainly devoted to marine hydrodynamics. The solver is based on a finite volume method to build the spatial discretization of the transport equations. The unstructured discretization is face-based. While all unknown state variables are cell-centered, the systems of equations used in the implicit time stepping procedure are constructed face by face. VOF approach is employed to handle free-surface. To enable relative motions of appendages, propellers or bodies, sliding and/or overlapping grid approaches have been implemented. An anisotropic adaptive grid refinement (AGR) procedure has been developed which is controlled by various flow-related criteria. It is also used to improve the accuracy of overset interpolations as it automatically smooths out the cell size distribution across overset interfaces between domains.

III – Numerical simulations

The test case is the well-known KLVCC2 tanker designed by KRISO in Korea. With the scale factor equal to 75, ship length Lpp = 4.267 m, draft T = 0.2773 m, beam B = 0.7733 m, ship speed U = 0.415 m/s, Fr = 0.0647. The radius of gyration are Kxx/B = 0.38 and Kzz/Lpp = 0.25. Zgc is located at 0.012m below free surface. Numerical simulations for turning circle and zigzag manoeuvres are compared with the measurement data obtained in the towing tank of Flanders Hydraulics (FH) and in that of MARIN with water depth to draft ratio h/T = 1.2. Although the same ship model was used, there are some differences between the two measurements. The width of the towing tank of FH is 7 meter, while that of MARIN is 15.8 meter. Due to the shorter length of the towing tank (68 meter instead of 220 meter), captive motion is imposed in the measurement of FH before the free run, while the ship model is accelerated on its own in the measurement of MARIN. For the turning circle manoeuvre, ship model follows a special path shown in Figure 1 in the measurement of FH. Moreover, rudder angle is set to 35° from the very beginning of the captive motion, while it is deflected from 0° to 35° when the free motion starts in the measurement in MARIN. Ship speed in the axial direction of the towing tank before the free run is the same in both measurements. Due to the oblique path applied during the captive motion stage in the measurement of FH, the ship speed is higher when the turning circle manoeuvre starts.

Numerical simulations have been performed by following the experimental procedure of FH with bank and confinement effects taken into account. For comparison, simulations have also been performed in a non-confined configuration. Computations performed in confined water contain three meshes: a background mesh representing the towing tank containing about 1M cells, an overset domain containing the ship with 4.5M cells with overset set boundaries located at 1.5m from the center line of the ship (more than 2B), and another overset domain for the rudder containing 1.7M cells. Similar mesh setup is applied for the non-confined water simulation. The only difference is that the width of the background domain is 3 times larger. Other settings for grid generation are similar to the setting in the previous studies for the KCS test case (Deng et al. [6]) and will not be reported here. Based on the previous studies for the KCS test case, such mesh resolution can provide accurate enough prediction. Hence, a grid independence study will not be performed in the present study. All simulations have been performed with the EASM non-linear turbulence model. Adaptive grid refinement is activated to ensure mesh size continuity at overset interfaces. A body force model based on open water propeller performance curve is employed to simulate the action of the propeller. The propeller revolution rate is determined from a self-propulsion simulation. The constant value thus determined is imposed for the manoeuvring simulation.

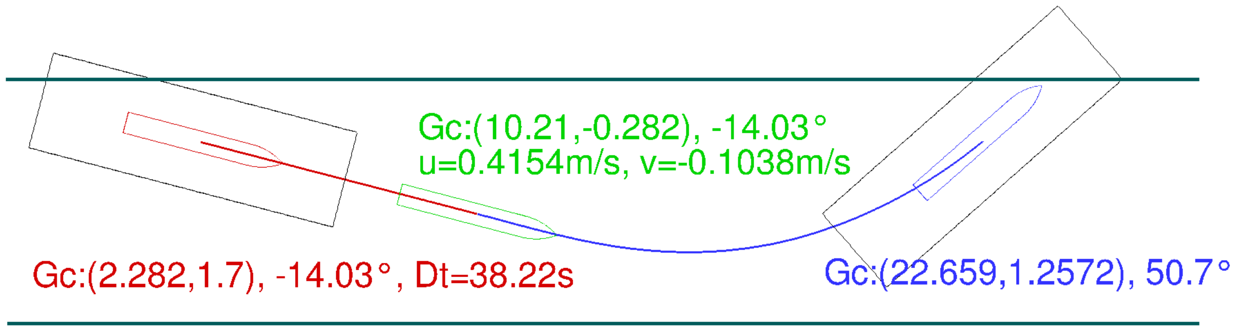


Figure 1. Setup for the TC35PS manoeuvre

CFD predictions for the TC35PS manoeuvre are compared with measurement data obtained by FH and by MARIN in Figure 2. As reported in [7], the repeatability of measurement results is not good for both experimental facilities. MARIN's measurement result shown in Figure 2 is one of the two measurement data that are in relatively good agreement with other two measurement results obtained by FH (not shown in the figure). The CFD simulation for the KCS test case in our previous study [6] reveals that when the symmetry of the flow is not well preserved during the captive stage, ship model turns faster in the turning circle manoeuvre. Since flow symmetry is better preserved in CFD simulation, CFD prediction is in closer agreement with the measurement result with larger gyration radius. Similar to the KCS test case, yaw rate is higher in CFD prediction in the confined water configuration. It is observed that the CFD prediction in the non-confined configuration is in better agreement with the measurement data obtained by FH. Ship model turns faster in model test by MARIN, especially for other test results not shown in the figure. We believe that it must be related to that fact that the ship model is not located in the centerline of the towing tank during the acceleration stage before the free run in the model test performed in MARIN. Hence, due to the bank effect, the yaw moment must be higher when that free run manoeuvre starts, resulting in a trajectory with smaller gyration radius.

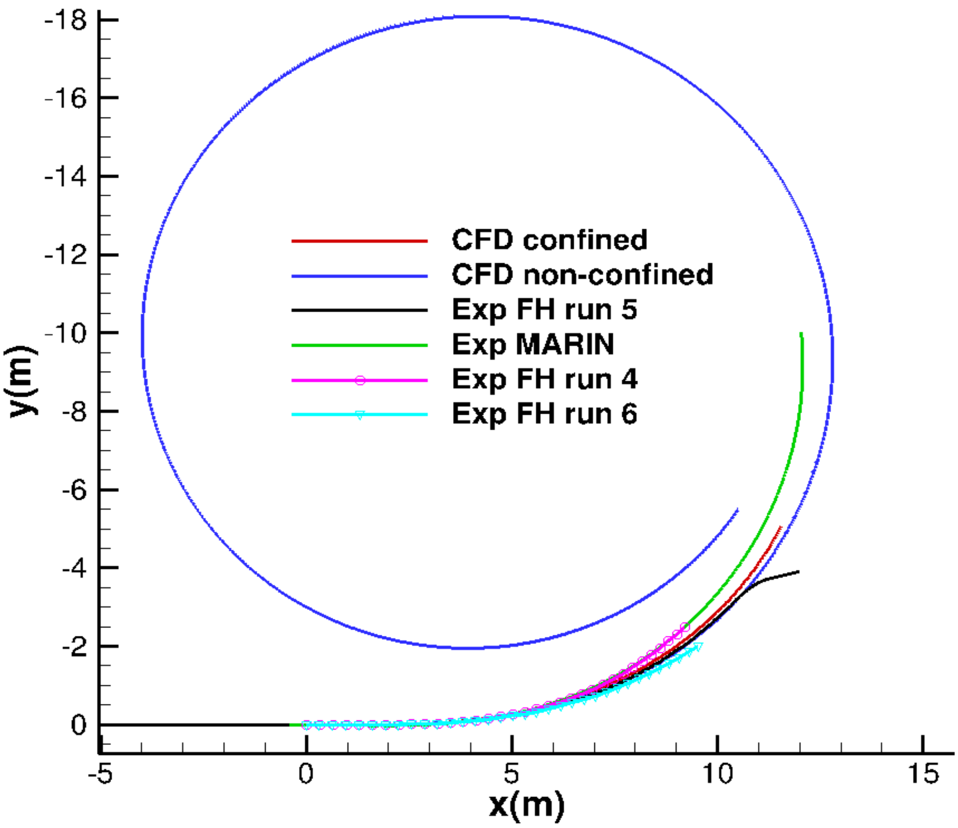


Figure 2. CFD predictions for the TC35PS manoeuvre

The predicted yaw rate for the TC35PS manoeuvre is compared with MARIN's measurement data and three repeated runs executed at FH in Figure 3. As observed in model test [7] in FH, the yaw rate drops quickly in the confined configuration when the ship approaches the bank near the end of the manoeuvre. The predicted yaw rate for the quasi-steady state agrees well with the measurement data. However, at the beginning of the turning circle manoeuvre, it is under predicted. We believe that it is possibly due to the bank effect, which is not taken into account in the simulation.

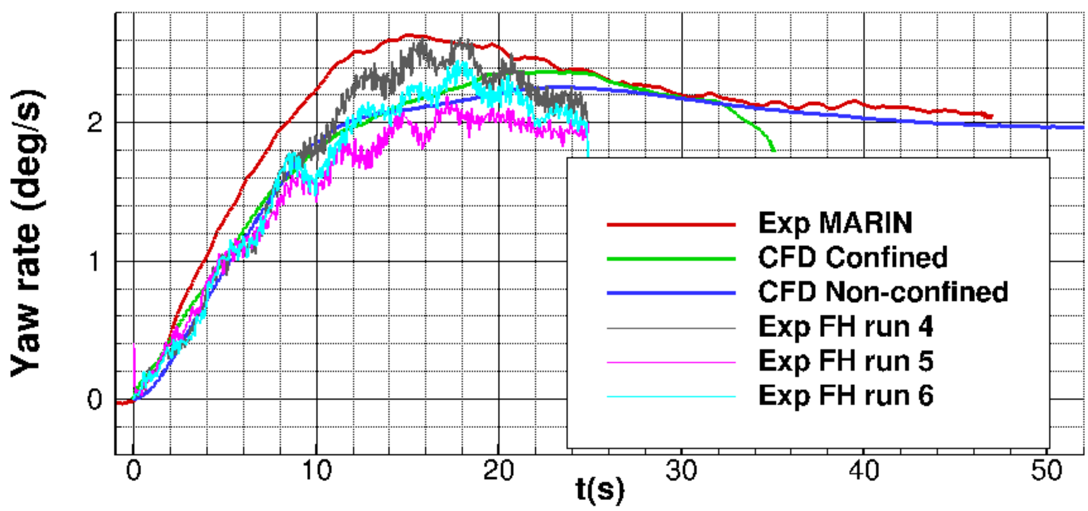


Figure 3. Yaw rate for the TC35PS manoeuvre

The yaw angle for the zigzag -20°/5° manoeuvre is compared with the measurement data obtained by FH in Figure 4. Rudder rate is 20°/s. The first overshoot angle is well predicted when the bank and the confinement effect are taken into account in the simulation. Later on, it is over-predicted by CFD simulation. The simulation performed in the non-confined configuration provides similar result with smaller overshoot angle.

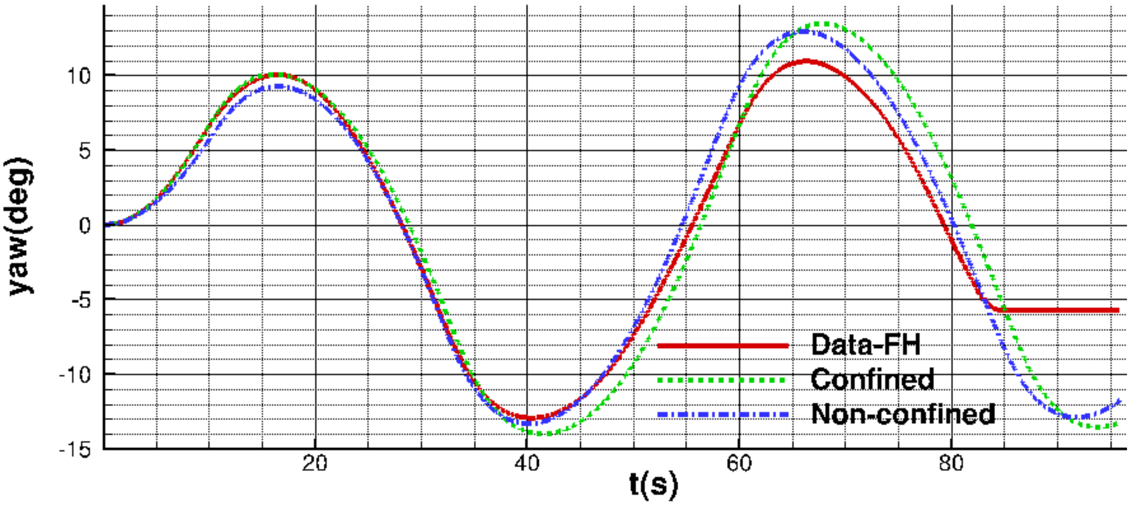


Figure 4. CFD prediction for the zigzag -20°/5° manoeuvre

The quality of the CFD prediction is confirmed by the result for the zigzag 20°/-5° manoeuvre shown in Figure 5. In this manoeuvre, the yaw angle is not zero when the free run zigzag manoeuvre starts in the model test performed by MARIN. This initial condition is taken into account in the CFD simulation. Moreover, the propeller revolution rate is adjusted from 7.16 rps for the previous to 7.66 rps so that the decay of ship velocity agrees better with the measurement data as shown in Figure 6. It is noticed that the rudder rate 20°/s applied in the simulation is the same as the measurement data. It is different from the value of 15.68°/s specified in the instructions of SIMMAN 2020 workshop for this test case. Bank effect is not taken into account in this simulation. The first overshoot angle is bigger compared with the case shown in Figure 4. It is equally well predicted by CFD simulation.

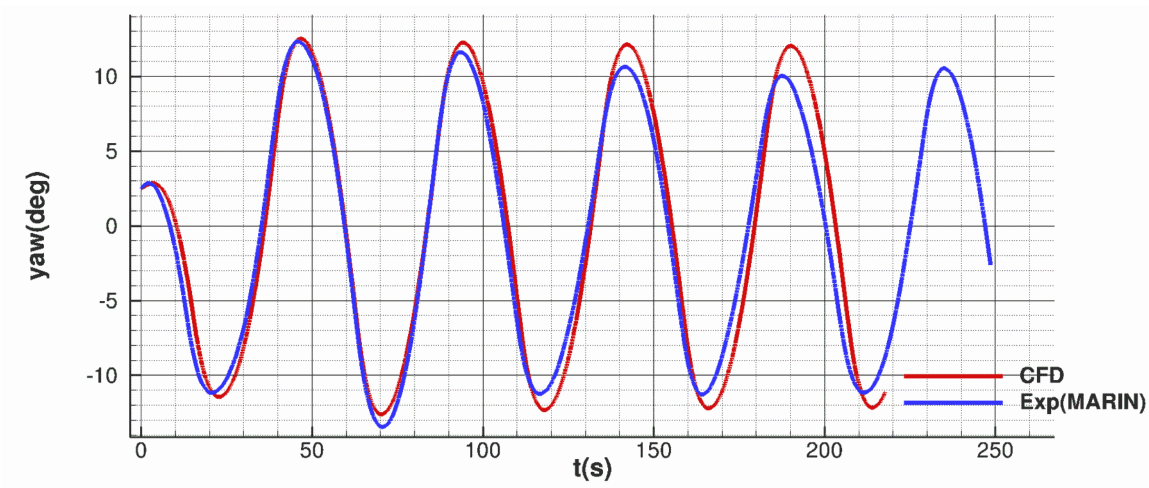


Figure 5. CFD prediction for the zigzag 20°/-5° manoeuvre

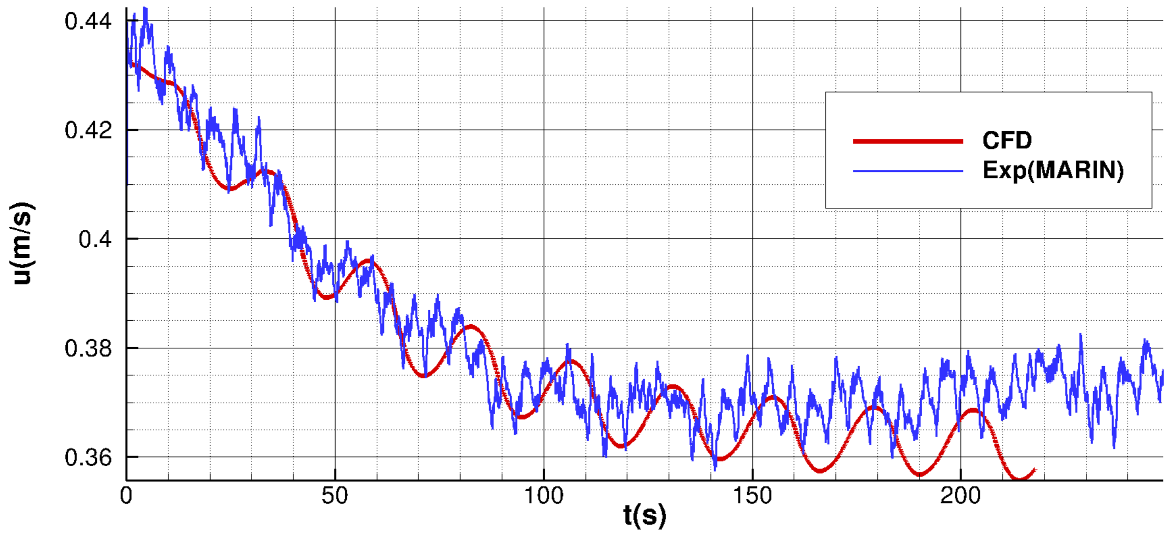


Figure 6. U velocity for the zigzag 20°/-5° manoeuvre

IV – Conclusions

The results presented in this paper confirm the finding of our previous study for the KCS test case, namely the accuracy of CFD simulation for ship manoeuvring in shallow water is similar to that in deep water. The confinement and bank effects are small for the zigzag manoeuvre in model test when the heading angle is reduced to 5°. It is not the case for the turning circle manoeuvre. Correct assessment of numerical simulation can be made only when the confinement and bank effects are taken into account in the numerical simulation. Moreover, it is also important to follow the experimental procedure before the free run manoeuvre starts in CFD simulation.

References

[1] Wang J., Zou L., Wan D., Numerical simulations of zigzag manoeuvre of free running ship in waves by RANS-Overset grid method [J]. Ocean Engineering 162 (2018) 55–79.

[2] Aram S., Mucha P., Computational fluid dynamics analysis of different propeller models for a ship manoeuvring in calm water [J]. Ocean Engineering 276 (2023) 114226.

[3] Deng G., Queutey P., Wackers J. et al. Assessment of ship manoeuvring simulation with different propeller models [J]. Journal of Hydrodynamics, 2022, https://doi.org/10.1007/s42241-022-0039-y.

[4] Carrica P., Mofidi A., Eloot K. et al. Direct simulation and experimental study of zigzag manoeuvre of KCS in shallow water [J]. Ocean Engineering 112 (2016) 117–133.

[5] Kim D., Tezdogan T., Incecik A. Hydrodynamic analysis of ship manoeuvrability in shallow water using high-fidelity URANS computations [J]. Applied Ocean Research 123 (2022) 103176. https://doi.org/10.1016/j.apor.2022.103176.

[6] Deng G., Eloot K., Hoydonck W. CFD simulation of turning circle and zigzag manoeuvre of KCS in shallow water. 第三十五届全国水动力学研讨会论文集, Aug. 2024, Harbin, China.

[7] Eloot K., Delefortrie G., Vantorre M. et al. Validation of ship manoeuvring in shallow water through free-running tests. Proceedings of the ASME 2015 34th International Conference on Ocean, Offshore and Arctic Engineering OMAE2015-41912, May 31-June 5, 2015, St. John's, Newfoundland, Canada.