DEVELOPMENT OF NUMERICAL TOOL FOR HYDRODYNAMICS SIMULATION OF HIGH SPEED PLANING CRAFTS

Ebrahim Jahanbakhsh
Marine Lab., Sharif Uni. of Tech., Tehran/Iran, ebrahim_jahan@yahoo.com
Roozbeh Panahi
Tarbiat Modarres Uni., Tehran/Iran, roozbeh_panahi@yahoo.com
Mohammad Saeed Seif
Dept. of Mechanics, Sharif Uni. of Tech., Tehran/Iran, seif@sharif.ir

SUMMARY

General purpose finite volume based computer software is developed to yield a time history of displacements, forces and moments, during the 6-DoF fluid-structure interaction in two phase flow. It uses a coupled VoF-fractional step method in solving the fluid flow and a boundary-fitted body attached hexahedral mesh in simulating the rigid body motions.

In this paper, the forward progress and the turning maneuver of a high speed planing catamaran is simulated. The results are analyzed and compared with the available data.

1. INTRODUCTION

Nowadays, numerical simulations are becoming a common way for assessment of ship performance in early design stages. Although model test using experimental approach is still very useful but has its own restrictions and expenses which has motivated to employ a numerical tool. Taking into account the advances in computer hardware, use of Computational Fluid Dynamics (CFD) is becoming the best choice in many cases.

In practice, a hydrodynamics problem includes turbulent viscous flow with complex free surface deformations and sometimes fluid-structure interaction. One of the practical ways to study the aforementioned coupled complicated case is to decouple it by either completely ignoring the less important phenomena or approximating them.

The motion of a floating or submerged body is a direct consequence of the flow-induced forces acting on it, while at the same time these forces are a function of the body movement itself. Therefore, the prediction of flow-induced body motions in viscous fluid is a challenging task and requires coupled solution of fluid flow and body motions. In recent two decades, with the changes in computer power, hydrodynamics motions simulation has been the subject of many numerical researches. Such studies started from restricted motions such as trim or sinkage and continued to evaluation of 6-DoF motions.

In this paper, the fundamental of a developed numerical tool which is capable of simulating the 6-DoF fluid-structure interaction is briefly presented. Then, a high speed planing catamaran is investigated in two cases of forward progress and turning maneuver. Discussion about the results is also included.

2. NUMERICAL TOOL

Here, a time dependent three-dimensional viscous free surface flow solver is implemented. The velocity and the pressure fields are coupled using fractional step of Kim and Choi. Over-relaxed and Gamma interpolations are used for the space discretization of the convection and the diffusion terms, respectively. One must take into account the presence of high density ratio phases e.g. water and air in discretization of the pressure integral which is treated in a new way. Also, a surface capturing method is used which solves a transport equation for calculation of fluids volume fraction. CICSAM interpolation has great advantages in comparison to other interpolations and used in space discretization of Volume of Fluid (VoF) transport equation. Also, the Crank-Nicholson interpolation is used in temporal discretization of all differential governing equations. More details are available in another paper of the authors to de develop a robust interfacial flow solver, Jahanbaksh et al. (1).

There are a variety of motion simulation strategies for numerical hydrodynamics applications such as deformable mesh, Chentanez et al. (2), re-mesh, Tremel et al. (3), sliding mesh Blades and Marcum (4), overlapping mesh, Carrica et al. (5), Cartesian mesh, Mittal and Iccarino (6), etc. Here, a hexahedral body-attached mesh following the time history of body motions is used. In other words, linear and angular momentum equations are solved in each time step which results in 6-DoF rigid body motions. Forces and moments of such equations are calculated by integration of normal and tangential stresses over the body surface as a result of flow solver. External loads can be also added to prepare the total forces and moments acting on the body. Such loads can be used to model the effect of rudder, thruster, mooring, etc. Resultant motions are then applied to the body as well as the mesh to make the computational domain ready for the next time step. It must be noted that, all of the fluid governing equations are written for a rigid control volume which moves with an arbitrary speed in the Newtonian Reference system. This feature which keeps the simplicity of the governing equations, results in using the relative face velocity for convection flux calculation taking into account the space conservation law. More details are presented in a recent paper by the authors, Panahi et al. (7).

The accuracy and the precision of the developed software (NUMELS-Numerical Marine Engineering Laboratory-Sharif) are strongly assessed in each stage of software development as shown in Table 1, Jahanbakhsh et al. (8), Panahi et al. (9), Jahanbakhsh et al. (10).

Table 1: Validation of the developed software

Case	Validation Problem
velocity-pressure coupling	orthogonal cavity flow
non-orthogonality	non-orthogonal cavity flow
volume fraction transport equation	scalar transport in the predefined constant oblique velocity field and Shear flow
two phase flow	Rayleigh-Taylor instability, dam breaking with and without obstacle, sloshing
wave generation and outlet boundary condition	Airy wave generation and transportation
6-DoF fluid-structure interaction	wedge and cylinder slamming, barge resistance and maneuvering, trimaran resistance

3. RESULTS

Now, the behavior of a high-speed planing catamaran shown in Fig.1 and Table 2, in forward progress and turning maneuver is evaluated.

The first step in all of the numerical simulations is to find an appropriate mesh. To simulate the catamaran, a wide variety of meshes is investigated and two of them are represented in Fig.2.

Anyway, after performing some study, an adequate mesh is found. The half domain of this mesh is shown in Fig.3 with the computational domain dimensions and the position of the craft.

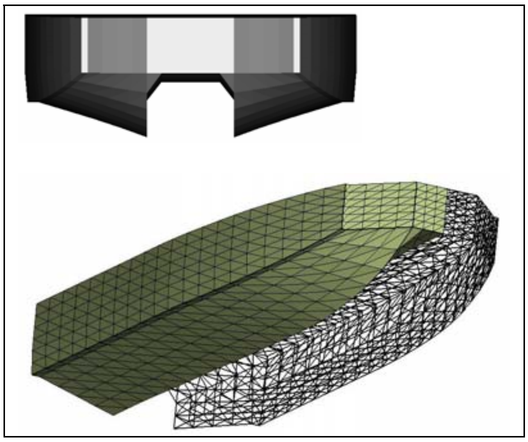

Fig.1: Catamaran geometry

Table 2: Catamaran ship characteristics

Characteristic	Value
length	12.3 m
width	4.6 m
Draft	0.45 m
Mass	17850 kg
vertical mass center position	0.25 m
longitudinal mass center position	3.81 m
Inertial moment around mass center	$\begin{bmatrix} 53274 & 0 & 0 \\ 0 & 295967 & 0 \\ 0 & 0 & 325563 \end{bmatrix}$

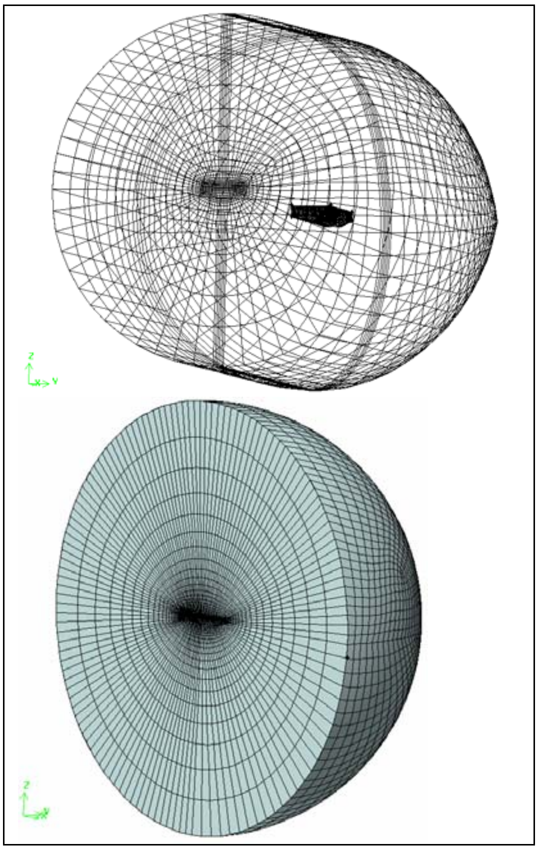

Fig.2: Two investigated catamaran mesh

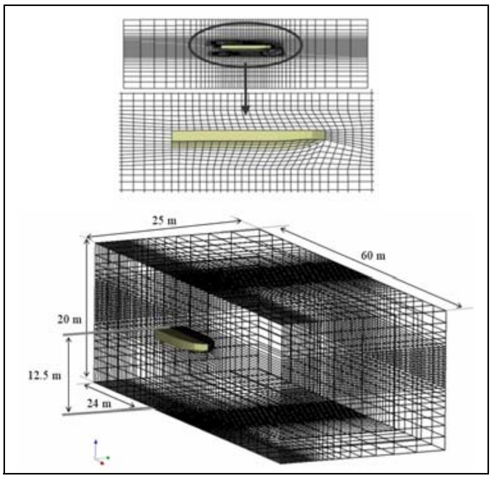

Fig.3: Catamaran appropriate mesh

3.1. Forward Progress

Forward progress in the case of the planing craft, is hardly affected by the changes in heave and pitch motions based on the hull form produced lift force. Considering the symmetry of the problem, a half domain with 95000 hexahedral cells is implemented. The thrust force is applied at 0.25 m under the mass center position, with two approaches of constant thrust and variable thrust.

In the constant thrust approach, a 40 kN force is exerted on the craft constantly from the initial time. In the variable thrust approach, an initially exerted 15 kN force is sharply changed to the next value just when an approximately steady behavior in forward progress is touched. In this approach, the examined forces are 15, 25, 30, 40, 45, and 50 kN. Such steps during 262 seconds of the simulation are presented in Table 3.

Table 3: Steps of changing thrust force

Step	Time Interval (s)	Thrust force (kN)
1	0.0-47.0	15
2	47.0-90.5	25
3	90.5-105.0	30
4	105.0-192.0	40
5	192.0-230.0	45
6	230.0-262.0	50

The time history of the results, using the second approach, is shown in Figs.4, 5, 6 and 7. As marked on the Fig.4, forward progress can be divided into three phases. In the first phase, which is from t = 0 s to t = 100 s, all diagrams behave smoothly. In this phase the craft is lifted about 0.2 m and its trim angle is increased up to 80. Velocity is about 10 kn at the end of this phase and experiences small changes except at the initial part of this phase. The second phase is between t = 100 s and t = 250 s. The distinct planing motion is occurred at the beginning of this phase during ten seconds, as it is obvious from the change in heave motion (Fig.5). In this phase, the craft is lifted about 0.55 m. The change in its trim angle is an interesting phenomenon because it is decreased from 80 to 40 in this phase, after an increase in the previous phase (Fig.6). Besides, the velocity is increased abruptly from 10 to 40 kn (Fig.7). The third phase of motion is accompanied by huge oscillations in all results. This is because of reaching an unstable dynamical position at the forward speed of 52 kn for this craft. Such a phenomenon which is accompanied with bow slamming is called propoising, and can be interpreted as a common case for such hull forms.

Fig.8 shows the plot of mean resistance versus velocity, extracted from Fig.4 and Fig.7. In this plot, the bold lines are curves fitted to result points. The left part of results belongs to 1st motion phase before planing occurrence. At this phase, the resistance experiences a 2nd order increase relative to forward speed. The right part of results belongs to 2nd and 3rd motion phases after planing occurrence. Here a 1st order increase of resistance is obvious. The dashed line which connects these two parts of results is an assumption which can be used as an estimate for the transient region. The gap is because of the fast increase in forward speed at the initial times of 2nd phase. Actually, there is no steady state position and therefore no resistance date in the mentioned interval. However, it is possible to cover this area with additional simulations.

Figs.9 and 10 show the comparison between numerical and experimental results of power and trim angle versus velocity, respectively. It is Obvious from Fig.9 that, the first approach (constant thrust) has a good performance in prediction of resistance and covers all velocities in contrast to the second approach (variable thrust). Besides, the results of the first and the second approach are near to each other. These two properties encourage using the first approach which is simpler in practice. The trim angle of the crafts is also plotted in Fig.10. It seems that using the second approach is better than the first approach in the case of trim angle, especially in evaluating its maximum value, although there is no point in that velocity.

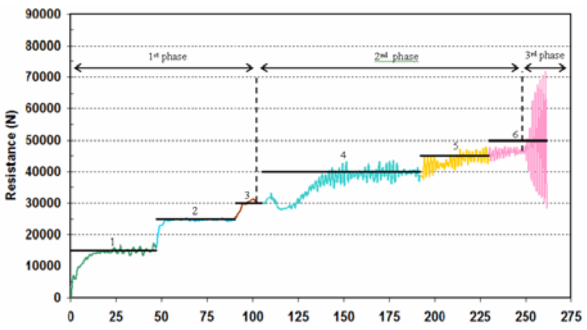

Fig.4: Resistance time history diagram using the variable thrust approach
(Bold lines represent thrust forces)

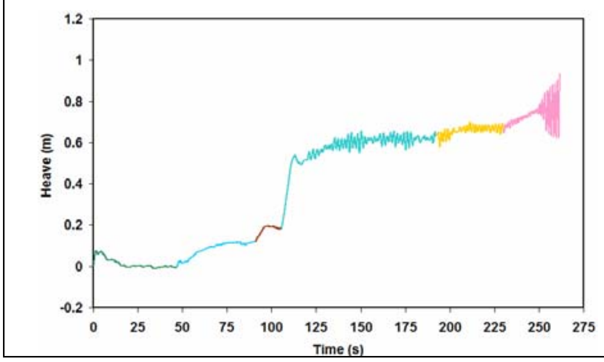

Fig.5: Heave motion time history

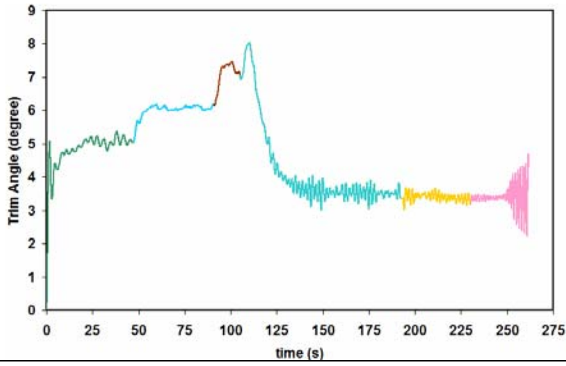

Fig.6: Pitch motion time history

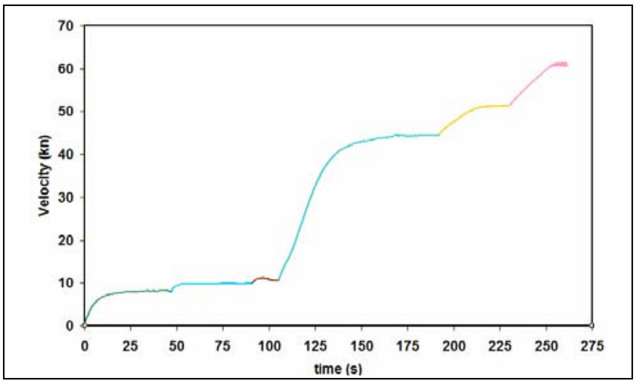

Fig.7: Velocity time history diagram

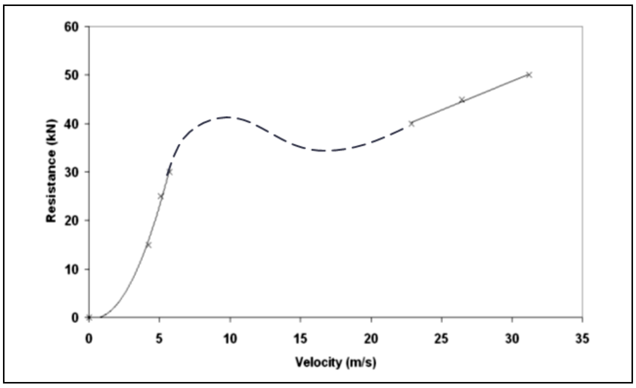

Fig.8: Resistance versus velocity

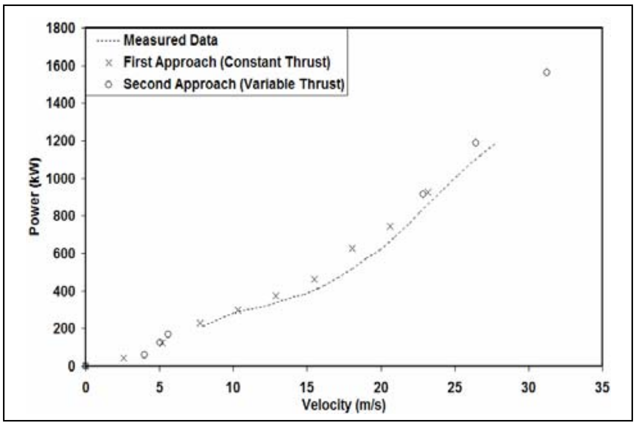

Fig.9: Numerical and experimental power

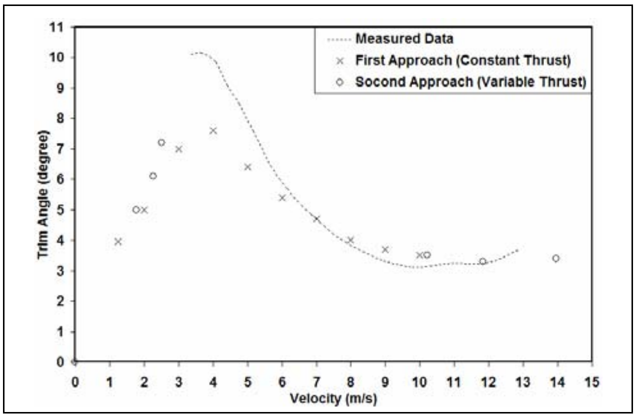

Fig.10: Numerical and experimental trim angle

Fig.11 shows some snap shots of the catamaran in different velocities. The depth of the water surface deformation at the stern of the craft is increased as the velocity is increased while its length is increased. The angle of the generated wave experiences a decrease in this manner. Wet-deck of the catamaran has different situation relative to water surface in different velocities. In low velocity and before planing the wet-deck becomes wet and in higher speeds it rises up from water as clearly represented in Fig.12.

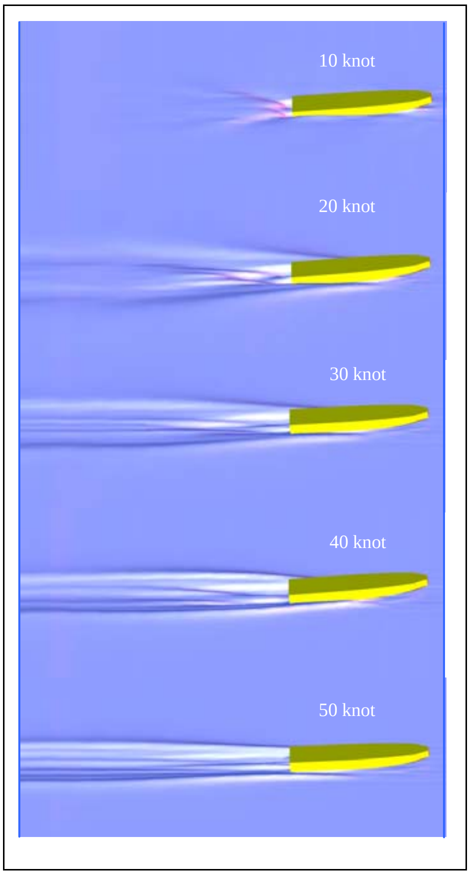

Fig.11: Snapshots and wave patterns of catamaran in different velocities

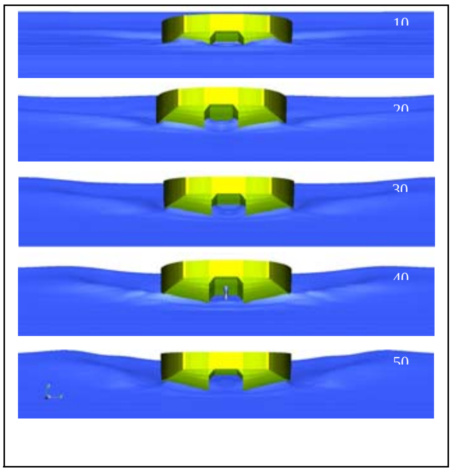

Fig.12: Front view of the catamaran

3.2. Turning Maneuver

Here the required force and moment of maneuvering are provided by apply a change in thruster angle relative to crafts longitudinal direction. Turning maneuver is simulated in two cases of 5 and 15 degrees. After 15 seconds from the beginning of the forward progress with 20 *kN* force, the thruster direction is changed to the mentioned angle. The time history of catamarans motions during the turning maneuver are presented in Fig.13.

Fig.13 (a) shows the time history of catamaran speed. It decreases until reaching a steady turning. It is obvious that, the difference between the forward (maximum) and the turning (minimum) speed and the gradient of speed change is increased as the thruster angle becomes larger.

Fig.13 (b) presents that the heel angle experiences a smooth behavior in the case of 5 degree thruster in comparison to 15 degree case which has a clear maximum value at the early stage of turning. Final trim of the catamaran is bigger in the case of 15 degree as could be predicted from the previous section. Also, yaw speed and drift angle have a same behavior during the tuning maneuver. Snapshots of catamaran are shown in Fig.13 (f).

Path of ship's center of gravity is shown in Fig.14. The turning circle and its diameter are decreased as thrusters' angle of rotation increased. Such behaviors are reasonable and qualitatively similar to experiment.

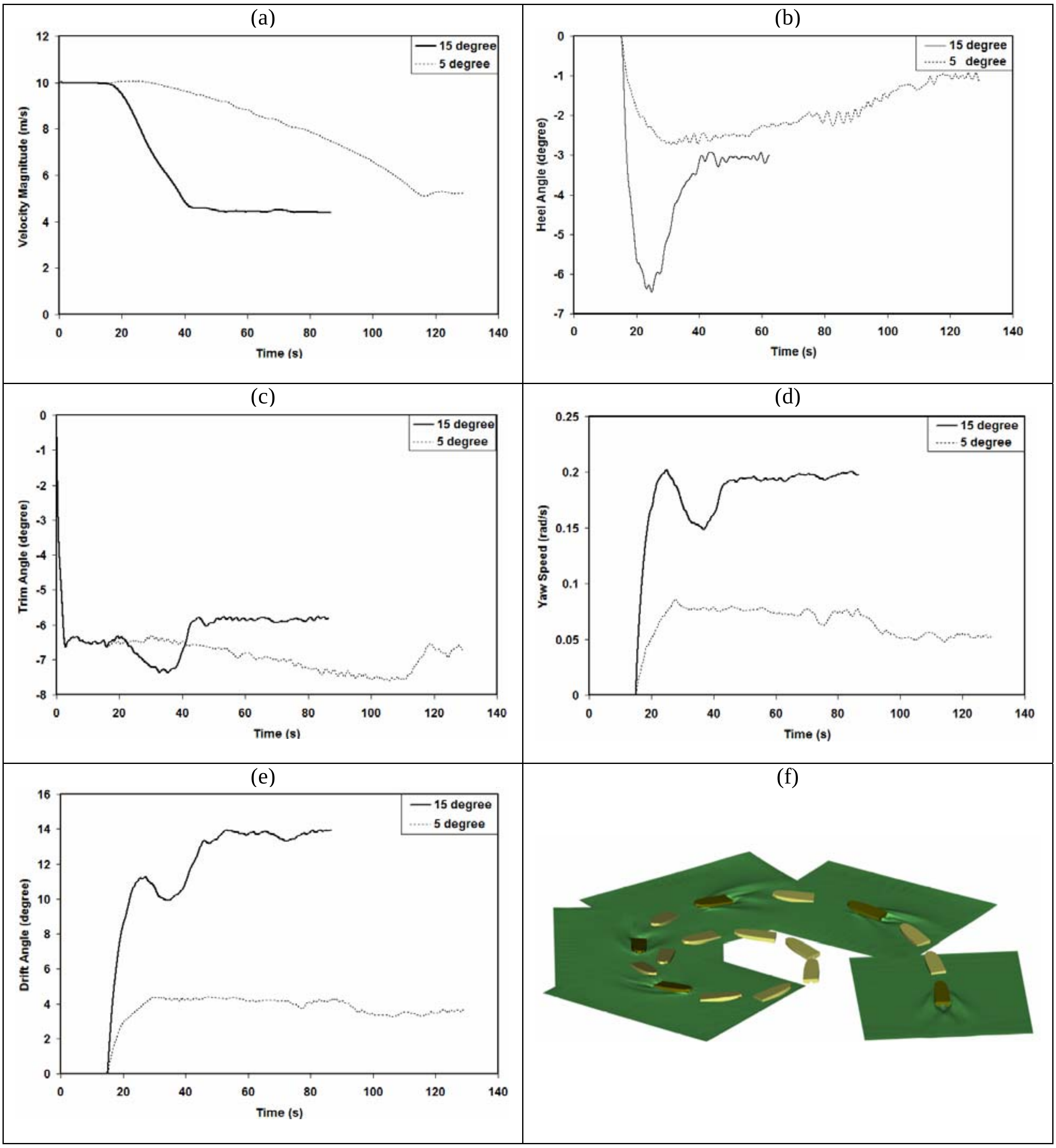
(a)
(b)
(c)
(d)
(e)
(f)

Fig.13: Catamaran turning maneuver time history

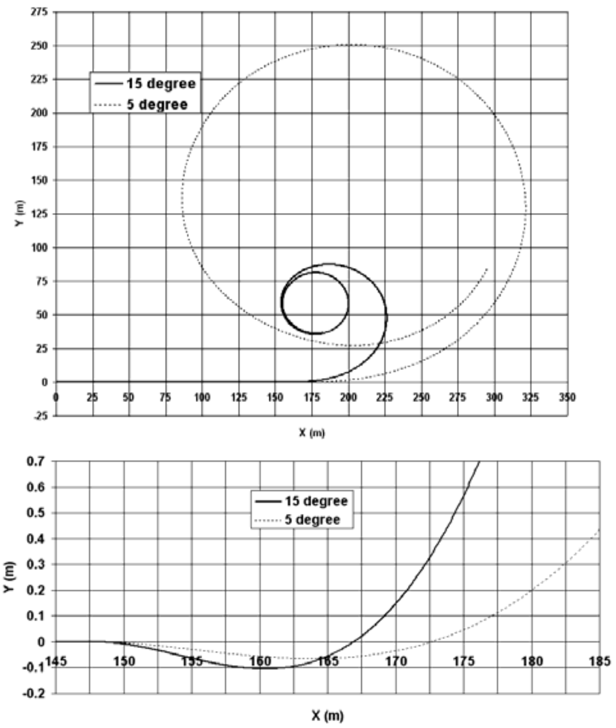

Fig.14: Ship mass center path, and overshoot

4. CONCLUSION

The proposed numerical algorithm is capable of simulating complex ship hydrodynamics problems. High speed catamaran investigated in this study is accompanied by some complicated phenomena such as planing and porpoising. However, the numerical results show a good agreement with experimental data in the case of forward progress. Besides, In the case of turning maneuver, the results are qualitatively acceptable. The presented computer software has no geometrical restriction and also an appreciable ability in a wide range of 6-DoF fluid-structure interaction including all types of crafts.

References

1. JAHANBAKHSH, E.; PANAHI, R.; SEIF, M.S. 2007, ‘Numerical Simulation of Three-Dimensional Interfacial Flows’, International Journal of Numerical Methods for Heat & Fluid Flow, 17(4):384-404

2. CHENTANEZ, N.; GOKTEKIN, T.G.; FELDMAN, B.E.; O’BRIEN, J.F. (2006), ‘Simultaneous coupling of fluids and deformable bodies’, Proceedings of the 2006 ACM SIGGRAPH/Eurographics symposium on Computer animation

3. TREMEL, U.; SORENSEN, K.A.; HITZEL, S.; RIEGER, H.; HASSAN, O.; WEATHERILL, N.P. 2007, ‘Parallel remeshing of unstructured volume grids for CFD applications’, Int. J. Num. Meth. Fluids, 53:1361-1379

4. BLADES, E.; MARCUM, D.L. 2007, ‘A sliding interface method for unsteady unstructured flow simulations’, Int. J. Num. Meth. Fluids, 53:507-529

5. CARRICA, P.M.; WILSON, R.V.; NOACK, R.W.; STERN, F., 2007, ‘Ship Motions using Single-Phase Level Set with Dynamic Overset Grids’, Computers & Fluids, doi: 10.1016/j.compfluid.2007.01.007

6. MITTAL, M.; ICCARINO, G. 2005, ‘Immersed boundary methods’, Annu. Rev. Fluid Mech, 37:239-261

7. PANAHI, R.; JAHANBAKHSH, E.; SEIF, M.S. 2006a, ‘Development of a VoF-fractional step solver forfloating body motions simuilation’, Applied Ocean Research, 28:171-181

8. JAHANBAKHSH, E.; PANAHI, R.; SEIF, M.S. 2005, ‘Multi-dimensional free surface flow simulation using two-phase Navier-Stokes solver’, 8th Numerical Towing Tank Symposium (NuTTs), Varna, 2-4 October

9. PANAHI, R.; JAHANBAKHSH, E.; SEIF, M.S., 2006b, ‘Development of a numerical hydrodynamic tank for ship motion simulation’, ECCOMAS CFD 2006, Egmond aan Zee, Netherlands, 5-8 September

10. JAHANBAKHSH, E.; PANAHI, R.; SEIF, M.S. 2006, ‘Ship Dynamic Simulation, Based on a Three-Dimensional Viscous Free Surface Solver’, 9th Numerical Towing Tank Symposium (NuTTs), Nantes, 1-3 October.