International Journal of Emerging Technologies in Engineering Research (IJETER)
Volume 4, Issue 10, October (2016) www.ijeter.everscience.org

Hydrodynamic Analysis of the Ship Hull

Ankush Kumar Srivastava
PG Scholar, Department of Mechanical Engineering, BTKIT, Dwarahat, India.

Hina Akhtar
PG Scholar, Department of Mechanical Engineering, BTKIT, Dwarahat, India.

Dr. Anirudh Gupta
Associate Professor, Department of Mechanical Engineering, BTKIT, Dwarahat, India.

Ravi Kumar
Assistant Professor, Department of Mechanical Engineering, BTKIT, Dwarahat, India.

Abstract – In this study the ship hull is model and simulated during its water operation in cruising. The CFD package used for this work is OpenFOAM. DTC Hull, which is a hull design of a modern 14000 TEU post-panamax container carrier hull is modelled using SOLIDWORKS, also only half part of the ship hull is modelled and for the remaining half symmetry boundary condition are applied to reduce the computational time. The mesh which is used in this work is generated using OpenFOAM mesh generation capabilities which include Block Mesh utility and Snappy Hex Mesh utility. InterDyMFoam solver is used to solve the multiphase flow equation and in addition to that Six DoF Rigid Body Motion solver is used for the mesh deformation. To handle free surface of fluid Multi-Dimensional Limiter for Explicit Solution (MULES) is used which implements Volume of Fluid Method (VOF). The turbulence model used in this work is k-ω Shear Stress Transport (SST) which comes under Reynolds-Average Navier- Stokes (RANS) model. The work in this study is to model and simulate the cruising of a ship by using dynamic mesh handling techniques provided in OpenFOAM. Also to show the fluid structure interaction during cruising and how it effects the different parameters of a ship hull.

Index Terms – Hydrodynamics, Hull, OpenFOAM, Dynamic Mesh, Finite Volume, CFD, Fluid-Structure.

1. INTRODUCTION

There are a number of challenges in computational modelling of the fluid structure interaction effects in the hull. These challenges occur mainly in modelling the multiphase flow along with complicated dynamic mesh motion and deformation. To attain a stable solution when using volume of fluid method for free surface modelling required a very small time step so that courant number remains in control. Due to the stated reason the computational time to run these simulations is very high for the complicated problems. There are only a few studies available in literature on computational fluid dynamics analysis of ship hull using OpenFOAM however, significant research literature exist on the design and analysis of ship and boat hulls using Ansys FLUENT.

In marine type vehicle, the designers generally work with two types of hull. The first is the most basic one i.e. displacement hull, which operates by plowing through water. It displace water to the sides and below during forward motion of the vehicle. The motion of water in this case is similar to that of an aerofoil moving in through the air which causes a low pressure field on the bottom surface of hull. Due to this negative pressure there is a suction force developed at the bottom surface. This creates a negative lift and also creates additional surface friction. Due to this there is a speed limit in displacement hull known as "hull speed".

The second type of hull is planning hull, which glides over the surface of water. This type of hull generates a lift while gliding over the surface. But in case of seaplanes the hull must have advantages of both displacement and planning boat.

Aktar has analysed drag forces in ship using CFD simulation because it is economically efficient. In his research 3-dimensional finite volume method has been applied to determine the drag coefficient. [2] For this analysis CFD software package FLUENT is used. Two turbulence model is used in this paper i.e. standard K-ε and Shear Stress Transport (SST) K-ω. With the help of the results of their CFD simulation, ship designer can choose minimum power with optimum speed and then can proceed to a model test for experiment results.

Garland has done Stepped Planing Hull Investigation. In his research he mentioned a vessel which is travelling with a high speeds enters in the planing regime of operation and how it experience hydrodynamic forces which generates lift to operate efficiently. [3.] He introduced a step in the hull at appropriate position and ensured that there is a flow separation after the step part. In his research he mentioned that there is substantial decrease in friction resistance because of flow separation which occurs because of the step.

Qiu and Song have done a decoupled aerodynamic and hydrodynamic analysis of amphibian aircraft during take-off. In their work they have divided the take-off process in number of small time steps and for each time step aerodynamics and hydrodynamics forces are calculated separately. For hydrodynamics force boat hull shape of amphibian aircraft is used with Volume of Fluid method. For aerodynamics they have used full take-off configuration.

The reason they have used decoupled method is because mesh which is used for hydrodynamic and aerodynamic calculation should be totally different to produce accurate results [4]. Another reason is that the minimum mesh size is different in aerodynamic and hydrodynamic drag calculation because hydrodynamic mesh use Volume of fluid method for which very small time step is used to kept courant number within limits.

2. PROBLEM FORMULATION

Geometry used in this work is of DTC Hull, which is a hull design of a modern 14000 TEU post-panamax container carrier. In this work only hull part of the ship is modelled. The software which is used for designing of hull part is SOLIDWORKS. Table 1 shows the details about of Hull.

Length of Hull	5.976 m
Width of Hull	0.859 m
Depth of Hull	0.244 m
Mass	825 kg
Surface Area	6.243 m^2
Volume	0.827 m^3
X	80
Y	1842
Z	1846

Table 1 Details of DTC hull.

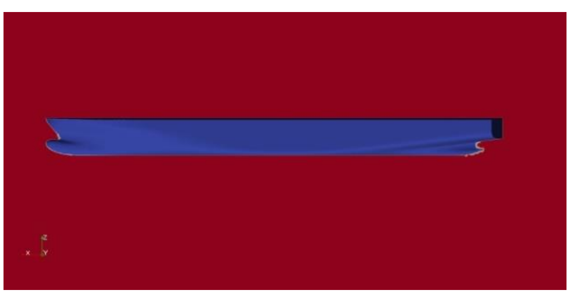

Figure 1 Geometry of DTC hull

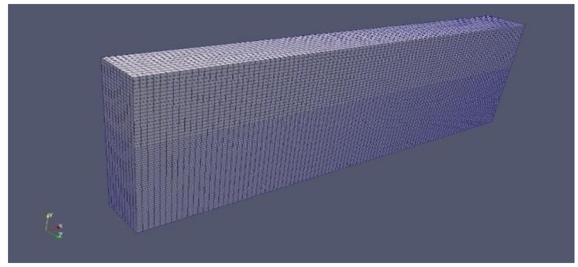

Figure 2 Domain is created using Block Mesh utility.

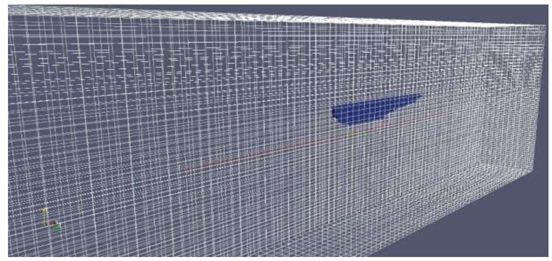

Figure 3 Surface Feature Extract utility extract STL file

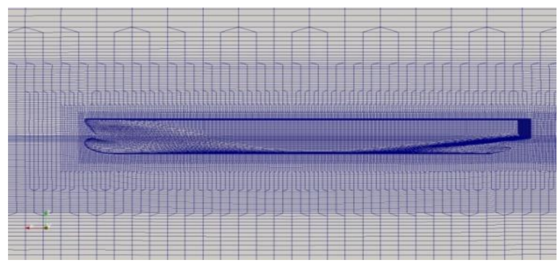

Figure 4 Hexahedral Mesh for DTC hull.

3. SOLVER

The solver which is used in this work is InterDyMFoam which is one of the solver in multiphase flows under CFD package OpenFOAM. This solver has dynamic mesh capabilities i.e. Six DoF Rigid Body Motion solver for the mesh deformation.

To solve the different type equation generated depending upon the problem InterDyMFoam use different type of numerical procedure or we can say solvers. The algorithm which is used in InterDyMFoam to solve Navier-Stokes Equation is Pressure-Implicit Method for Pressure linked equation (PIMPLE). PIMPLE is a combination of Pressure-Implicit with Splitting of Operators (PISO) and Semi-Implicit Method for Pressure Linked Equation (SIMPLE).

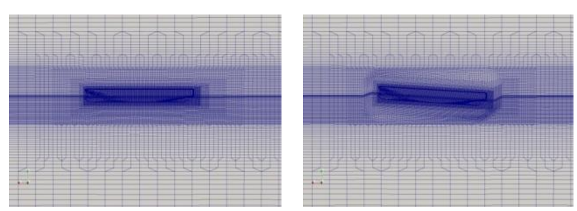

Figure 5 Dynamic Mesh deformation.

4. RESULTS AND DISCUSSION

Cruising-In cruising there are three cases where the hulls are tested at three different velocity of 1.67 m/s, 2.5 m/s, 3.0 m/s respectively. In all three cases dynamic mesh is used for simulation work and pitching is allowed.

Case 1 (velocity 1.67 m/s with pitching)

In this case of cruising, the hull is tested at velocity of 1.67 m/s. In this case dynamic mesh is used for simulation work and pitching is allowed. The Contour plot for alpha water where alpha water = 1 shows the presence of water.

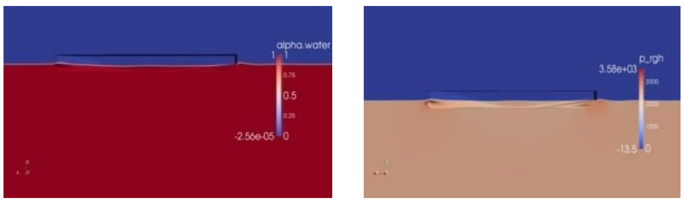

Fig 6(a) Contour plot for alpha.water of 1.67 m/s cruising Fig 6(b) Pressure variation contour of ship hull at 1.67 m/s

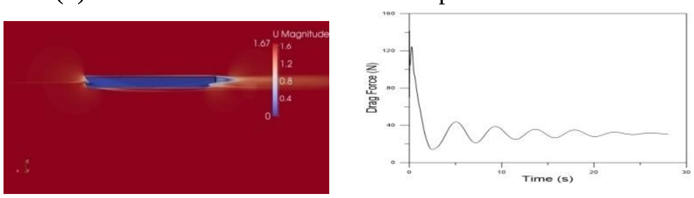

Fig 7 (a) Velocity variation contour along the length of ship hull at 1.67 m/s Fig 7 (b) Variation of Drag Force with Time and initial velocity of 1.67 m/s with pitching

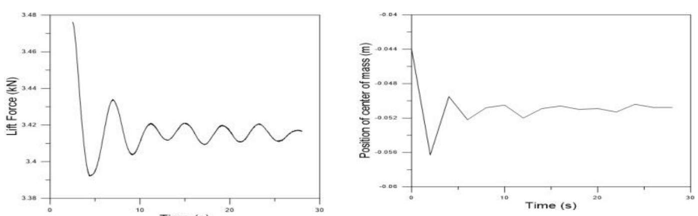

Fig 8 (a) Variation of Lift Force with Time and initial velocity of 1.67 m/s with pitching. Fig 8 (b) Variation of center of mass with Time and initial velocity of 1.67m/s with pitching.

Case 2 (velocity 2 .5 m/s with pitching)

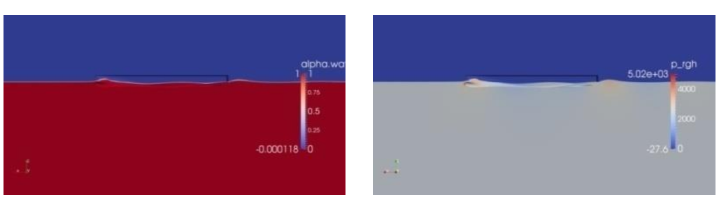

Fig 9 (a) Contour plot for alpha water for 2.5 m/s cruising. Fig 9 (b) Pressure variation contour along the length of ship hull at 2.5 m/s

In this case of cruising, the hull is tested at velocity of 2.5 m/s. In this case dynamic mesh is used for simulation work and pitching is allowed.

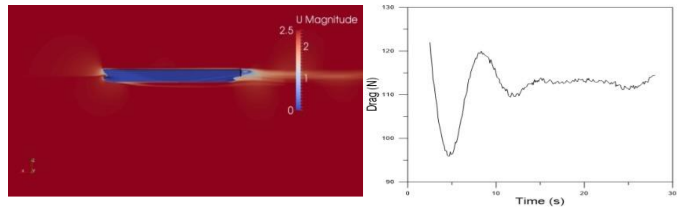

Fig 10 (a) Velocity variation contour along the length of ship hull at 2.5 m/s Fig 10 (b) Variation of Drag Force with Time and initial velocity of 2.5 m/s with pitching.

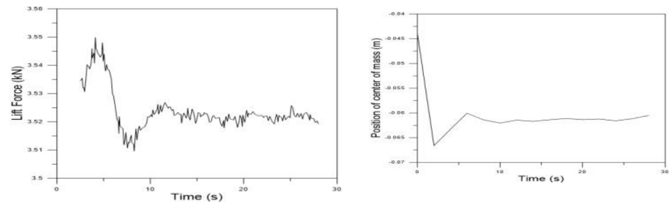

Fig 11 (a) Variation of Lift Force with Time and initial velocity of 2.5 m/s with pitching. Fig 11(b) Variation of Center of mass with time and initial velocity of 2.5 m/s with pitching.

Case 3 (velocity 3 .0 m/s with pitching)

In this case of cruising, the hull is tested at velocity of 3.0 m/s. In this case dynamic mesh is used for simulation work and pitching is allowed.

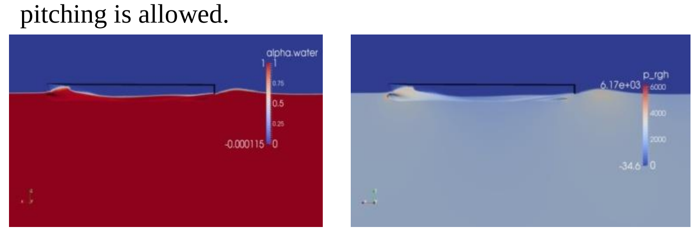

Fig 12 (a) Contour plot for alpha water for 3.0 m/s cruising Fig 12 (b) Pressure variation contour along the length of ship hull at 3.0 m/s

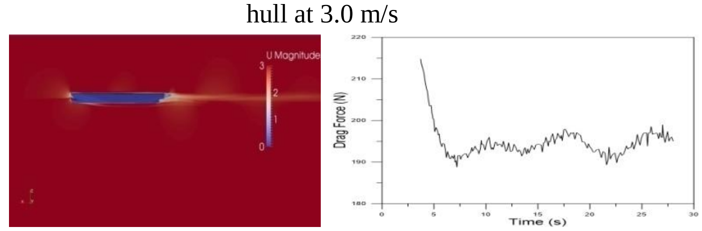

Fig13 (a) Velocity variation contour along the length of ship hull at 3.0 m/s Fig 13(b) Variation of Drag Force with time and initial velocity of 3.0 m/s with pitching.

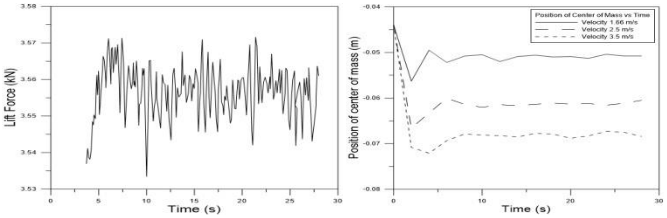

Fig 14(a) Variation of Lift Force with Time and initial velocity of 3.0 m/s with pitching. Fig 14 (b) Variation of Center of mass with Time and initial velocity of 3.0 m/s with pitching.

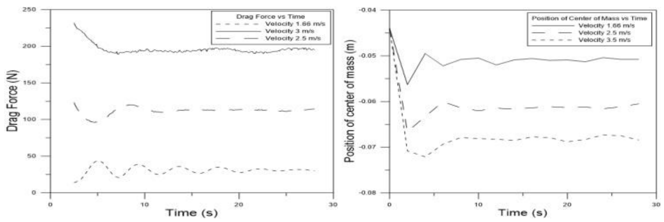

Fig 15 (a) Comparison of Drag Force for hull at different velocities. Fig 15 (b) Comparison of center of mass for hull at different velocities.

5. CONCLUSION

The hydrodynamic analysis of ship hull at different velocities was analyzed and following conclusions were made:

- In Cruising the hull is tested at velocity of 1.67 m/s, 2.5 m/s and 3.0 m/s respectively. In this case dynamic mesh is used and pitching is allowed.
- Higher the cruising velocity more will be the drag force.
- The cruising velocity doesn't have much effect on lift force.
- Higher the velocity more will be the suction pressure developed at the bottom.
- Due to more low pressure at high speed, position of centre of mass is lower.
- The main advantage of OpenFOAM is that it gives you a platform where you can implement your own codes.

REFERENCES

[1] Husa, Bill., " Stepped Hull Development for Amphibious Aircraft" orion Technologies Aerospace Design And Engineering 28 june (2000).

[2] Aktar, S., Saha, K.G. and Alim, A.Md., " Drag Analysis of Different Ship Models Using Computational Fluid Dynamics Tools" The International Conference on Marine Technology 11-12 December 2010, BUET, Dhaka, Bangladesh.

[3] Garland, R.W., " Stepped planing Hull Investigation" United States Naval Academy.

[4] Qiu, l. and Song, W., " Efficient Decoupled Hydrodynamics and Aerodynamics Analysis of Amphibious Aircraft Water Take-Off Process" , Journal of Aircraft Vol.50, No. 5, September-October 2013.

[5] Ertinaz, F., " Computational Analysis of Multiphase Ship Resistance including 6-DoF Motion using OpenFOAM", 2nd International Symposium on Naval Architecture and Martime, November 7, 2014.

[6] Koch, M., Lechner, C., Reuter, F., Kohler, K., Mettin, R. and Lauterborn, W., " Numerical modelling of laser generated cavitation bubbles with finite volume and volume of fluid method, using OpenFOAM", Journal of Computers and Fluids, Volume 126, pp. 71-90, March (2016).

[7] Mahady, k., Afkhami, S. and Kondic, L., " A volume of fluid method for simulating fluid/fluid interface in contact with solid boundaries" , Journal of Computational Physics, Volume 294, pp. 243-257, August (2015).