The 10th International Conference on Coasts, Ports and Marine Structures (ICOPMAS 2012)
Tehran, Iran, 19-21 Nov. 2012

Investigation on Resistance of a Bulk Carrier Vessel Using CFD Method by VOF model

[aboozar . ebrahimi ابوذر ابراهیمی]
[محسن خبیر mohsen . khabir]

Key Words: *Bulk Carrier Vessel, Resistance, CFD, FLUENT, VOF Model, Experiment.*

1- Introduction

Bulk carriers have an important portion of world maritime transport. The world's bulk transport has reached immense proportions: in 2005, 1.7 billion metric tons of coal, iron ore, grain, bauxite, and phosphate were transported by bulk carrier ships [1]. Today, the world's bulker fleet includes 6,225 ships of over 10,000 tons deadweight and represents 40% of all ships in terms of tonnage and 39.4% in terms of vessels [2].

In recent years, the numerical solution method and specially Computational Fluid Dynamics (CFD) have developed for studying hydrodynamics of ships. The CFD method in comparison with model test has great advantages such as lower costs, accessibility and more visible details in results e.g., pressure and velocity contours, vectors and gradients. One of the most important weaknesses of this method is less accuracy than that of model test. Therefore, experimental results should be used for verification of CFD results. In this study, we use CFD method for calculating resistance of a bulk carrier model and then verify results by experimental data from model test.

Numerical study on resistance of a ship has been performed in several cases. Thornhill et al. [3] used a finite volume code to simulate the flow around a planning vessel at steady speed through calm water using 3D unstructured hybrid mesh. Force and moment data from the simulations were used in an iterative scheme to determine the dynamic equilibrium position of the model at selected speeds. Van et al [4] measured flow around a VLCC tanker model. Ogiwara et al. [5] carried out a study on series 60 and calculated pressure distribution around hull of that. Jones et al. [6] used FLUENT code for numerical simulation of flow around a modern naval ship, DTMB 5415 and calculated waves and free surface of it. Obreja et al [7] carried out a series of model tests in a bulk carrier vessel. In this study, we use results of these tests for verification of numerical results. Accordingly, we used the geometry of experimental model for numerical modeling.

Banks et al [8] calculated components of container ship resistance numerically. They used ANSYS CFX for numerical simulation and compared CFD results with experimental results. Hakan et al [9] modeled a ship hull by using FLUENT commercial code and calculated resistance of model. Finally they compared numerical results with experimental data.

2-Model Geometry

A model of a Panamax bulk carrier ship is used in this study that is a 1:80 scale of full scale ship. The bulk carrier model is shown in Fig. 1. The characteristics of the ship and of the model are presented in Table 1.