Numerical Simulation of Flow around a Podded Propeller

Wei Li1 and Chi Yang2
1 School of Naval Architecture, Civil and Ocean Engineering, Shanghai Jiao Tong University
Shanghai, China

2 Dept. of Computational and Data Sciences, George Mason University
Fairfax, Virginia, USA

ABSTRACT

In this paper the viscous steady flow around a podded propeller is simulated using two CFD tools. Specifically, an open source CFD toolbox, called OpenFOAM, is further developed to study the open water hydrodynamic performance of the podded propeller and the hydrodynamic interactions between the pod and the propeller. The standard k-ε turbulence model is used in the viscous flow simulation. In addition, the same simulation studies are also performed using commercial CFD software FLUENT. Numerical results obtained using two CFD tools are compares with experimental measurements. Fairly good agreement is obtained. The interaction between the strut and the propeller and the effect of the strut on the hydrodynamic performance are analyzed by comparing the flows around the podded propeller and the single propeller.

KEY WORDS: Podded-propeller; OpenFOAM; viscous flow.

INTRODUCTION

Podded propellers have recently been widely used in modern commercial ships due to their relatively high efficiency and good sea-keeping ability. As a new type of propulsion, podded propellers usually have large hubs and good maneuverability. Podded propellers can reduce noise, vibration and the fuel consumption of a vessel with the proper design of the pod and the strut. The overall efficiency of a podded propulsion system can have a gain in the order of 2% to 4% over a conventional propeller. In addition, podded propellers can be used in dynamic positioning systems of floating structures to provide thrust in all horizontal directions by being mounted on the bottom of the hull.

The performance of the podded propeller has been studied numerically and experimentally. For example, Mewis (1998) investigated the open water performance of podded propellers with different configurations. The test was performed to study the total propeller thrust and the total unit thrust, including scale effects leading to an accurate power prediction. The effective wake fraction and the blade thrust were also provided in this study. Poustoshniy et al. (1998) investigated the scaling problems of the pushing-mode podded propeller.

The numerical methods based on potential flow theory have been used widely and successfully to predict the performance of conventional propellers. The modified wake model must be assumed in order to predict the performance of podded propellers using potential-flow based numerical method. The potential-flow method was used to calculate the flow around podded propellers by Ghassemi and Allievi (1999), Yang and Ma (2003) and Ma et al. (2006) with a modified wake model. However, different wake models will affect the accuracy of the prediction directly. With the development of computer hardware and the numerical methods based on Euler/RANS solvers, many difficult and complex flow phenomena can be modeled directly, including the detailed flow information in the wake of the flow around a propeller. Kinna et al. (2004) applied the coupled Euler solver with a potential flow method to analyze the performance of podded propellers. Sanchez-Caja et al. (1999) and Li et al. (2007) applied viscous flow methods to study the flow around a podded propeller.

OpenFOAM (Open Field Operation and Manipulation) is an object oriented C++ toolbox for solving various systems of partial differential equations using the finite volume method on arbitrary control volume shapes and configurations. It includes preprocessing (grid generator, converters, manipulators, case setup), postprocessing (using OpenSource Paraview), and many specialized CFD solvers (http://www.openfoam.org). The features in OpenFOAM are comparable to what is available in the major commercial CFD codes. Some of the more specialized features that are included in OpenFOAM are: sliding grid, moving meshes, two-phase flow and fluid-structure interaction. Since OpenFOAM is an open-source code, it is possible to gain control over the exact implementations of different features and it is reasonably straightforward to implement new models and fit them into the whole code structure. Many researchers are using OpenFOAM, which allows international exchange of development.

Based on the successful application of OpenFOAM in the simulation of the flow around a conventional propeller, the OpenFOAM is further developed in this study to investigate the open water hydrodynamic performance of the podded propeller. The open water performance results are compared to those measured in experiments and obtained by other software. The effect of strut on the performance of the podded propeller is analyzed and the hydrodynamic interactions between the pod and the propeller are studied in the present paper.