An Optimization Process for Propeller Design and its Application Based on CFD

Jeong-yong Park1, Milovan Peric2, Daehwan Park3

1Hyundai Heavy Industries, Co., Ltd.
Ulsan, Korea
2CD-adapco
Nurnberg, Germany
3FRIENDSHIP SYSTEMS AG
Potsdam, Germany

ABSTRACT

In the present study, optimization of propeller geometry for a merchant ship was mainly studied. In this regard, simulations based on RANS-equations were conducted to predict the self-propulsion performance. The numerical simulation was automatically performed by JAVA macro using the commercial CFD software, STAR-CCM+. The commercial optimization software CAESES(former name : FRIENDSHIP-Framework) was integrated with the flow solver STAR-CCM+ in order to achieve an automatic optimization process. Through the optimization study, the relationship between propeller geometry and self-propulsion performance was investigated systematically. Understanding the characteristics of propeller shape based on the view point of self-propulsion provides good chances to establish the new guideline of propeller design.

KEY WORDS:  Propeller; Optimization; CFD; STAR-CCM+; CAESES; Propeller-hull interaction; DOE

INTRODUCTION

With the implementation of EEDI, energy saving has attracted a wide attention in ship design. These days, advances in computer power are revolutionizing design process for all ship components (hull, propeller, rudder, appendages). Computational Fluid Dynamics (CFD) is allowing propeller designers to develop efficient propellers optimized to improve the propulsion performance by considering hull-propeller-rudder interaction. As a result, propeller design is addressed again as an area of increasing focus to save energy.

On the development of propeller design method, Betz (1919) introduced lifting-line method and then Goldstein (1929) and Lerbs (1952) improved it. Later on, computer program based on lifting surface vortex lattice method began to be utilized (Greely and Kerwin, 1982). In last decades, propeller design has been conducted primarily with numerical tools based on potential flow theory or by designer’s experience in a trial and error manner. Potential solvers are very useful to predict perceptive approximation of hydrodynamic performance in a short time. However, they are not suitable to consider hull and rudder geometry or interaction between hull and propeller. Due to these reasons, propeller designers have trouble in designing an improved propeller which is better than the reference (stock) propeller. It is necessary to consider the interaction between propeller and hull (and other appendages) in three dimensions (3D) for optimizing the total propulsion efficiency.

There are several earlier studies (Lee, 2004; Jung, 2007) concerning propeller optimization. In most of them, programs based on potential flow theory were used. The 3D hull, rudder geometry and their interaction with propeller were not taken into account in those computations. With the advance in computing power, several propeller optimization studies that account for the interaction between hull and propeller were recently published. Vesting and Bensow (2011) treated the flow near hull with a viscous Reynolds-averaged Navier-Stokes (RANS) methodology, using SHIPFLOW, coupled to an inviscid potential program MPUF-3A which replaces propeller effect.

In our optimization research, self-propulsion computation with detailed 3D geometry of hull, rudder and propeller is fully performed by CFD based on RANS-equations, in order to account for the hull-rudder-propeller interaction as accurately as possible.

NUMERICAL CALCULATION DESCRIPTION

The numerical simulation and post processing of self-propulsion using STAR-CCM+ is conducted automatically, controlled by a JAVA macro for integration with commercial optimization software, CAESES. First, a one-off self-propulsion simulation was conducted for a reference propeller. The converged solution is used as an initial solution for the simulation of another propeller generated in the optimization process. This not only reduces the computing time but also allows an easy comparison with reference propeller. For the calculation of self-propulsion, the blade geometry generated by CAESES is imported into STAR CCM+.

In this study, the grid consists of two parts (Fig. 1): one is attached to propeller and rotates with it as a rigid body (region B), while the other is attached to the hull and is stationary (region A). The two grids have a cylindrical sliding interface. For the simulation with all additional propeller designs, only meshes in region B are re-generated while the mesh in region A is maintained. The general information for the numerical calculation is presented in Table 1.