Numerical Flow Analysis of a 3-D Hydrofoil Using a Surface Panel Method

Riccardo Bulgarelli, Decheng Wan
School of Naval Architecture, Ocean and Civil Engineering. Shanghai Jiao Tong University
Shanghai, China

ABSTRACT

This paper presents the application of a CFD tool for the flow analysis of marine 3-D hydrofoils. A boundary element panel method is used to solve the potential flow past a typical ship rudder, and obtain the forces acting on it. Investigations are carried out on the optimal settings, in order to tailor the method for this type of analysis. A comparison of the results is done both with experimental data and with RANSE computations, to validate the code and to determine its accuracy and limits. The comparison shows the great efficiency of this method, which gives precise results in a minimal computational time.

KEY WORDS: Hydrofoil, Potential Flow, Boundary Element Panel Method, Marine Rudder.

INTRODUCTION

In the preliminary hydrodynamic and structural design of marine hydrofoils the main task is to analyze the magnitude of the forces and moments acting on it. At this stage it is important to have a fast and reliable tool because a wide range of investigations has to be carried out in a short time. The variety of cases to be tested depends on the many possible geometrical configurations, and on the different inflow conditions to be tested. The same requirements are shared also by other fields of ship design, for example in shape optimization algorithms, where a large number of computations as to be recursively performed in a fairly short time and without losing rating capabilities.

In this paper it will be shown that the presented boundary element surface panel method is an optimal CFD tool to suit the above mentioned purposes. In particular, a well-known formulation of this numerical technique will be implemented and its main settings investigated in order to tailor it for the solution of the flow past a hydrofoil. A practical benchmark case (a marine rudder) will used to show the good accuracy, together with fast computations, of the presented methodology for this application. Therefore it will be demonstrated that this CFD tool can be used efficiently in the design of typical marine hydrofoils, such as rudders, sailing yacht keels and roll fin stabilizers.

Boundary element surface panel methods are a numerical technique to solve the irrotational inviscid fluid flow around a body. The formulation is based on the solution of the Laplace equation through the superposition of fundamental solutions (source, dipole or vortex) distributed over small panels (most often quadrilaterals), which discretize the surface of the body.

Panel methods have been successfully developed since the 1960s (for non-lifting bodies), mainly for aerodynamic applications in the aircraft industry. The low requirements of computational power favored their early development, which is however continued until recent years as they still remain a very efficient CFD tool. A hydrofoil is a three-dimensional lifting body moving in a uniform infinite fluid. The first formulation of the panel method successful in solving this type of problem was developed by Hess (1972); it was characterized by the use of constant strength source and dipole singularities. This pioneering work represented the basis for most of the following developments and its approach is largely followed also in the work presented in this paper. Several variations of this method have then been proposed, above all attempts to create higher fidelity models: using higher orders singularities (e.g. Johnson, Ehlers and Rubbert, 1976), iterative wake roll-up models (e.g. Fiddes and Gaydon, 1996) and panel methods coupled with boundary layer theory (e.g. Maskew, 1987). Although reaching higher degrees of accuracy, these methodologies generally require also a much higher computational time and may be subject to poor stability. These two issues not always fit the above mentioned requirements of a CFD tool for preliminary design or for use in optimization algorithms, as we address in this work.

In the field of naval architecture the use of panel methods started later, but then gained remarkable popularity. The main applications have been in the computation of ship free surface flows (Dawson, 1977; Yeung, 1982), flow around a propeller and propeller-rudder interactions (Turnock, 1993), rudder fluid-structure interactions (Turnock and Wright, 2000).

The available experimental data about hydrofoils in free stream flow are rather limited; a useful contribution is provided by Molland and Turnock (1991). These authors carried out wind tunnel investigations of a few rudder models normally used on commercial vessels. In this