

## CFD application for turbulent flow analysis around the sails and hull of a sailing yacht

Wu-Joan Kim and Hye-Ryouon Chi

Department of Naval Architecture & Marine Engineering, Mokpo National University  
Muan-gun, Jeonnam, Korea

### ABSTRACT

CFD tools were used for turbulent flow analysis around a sailing yacht. Turbulent flows are solved around a sail system composed of main and jib sails with a mast to assess the sail performance and to determine the center of effort. Flows are also solved around a sailing yacht hull with keel and rudder with and without free surface effect. Both calculated results for the sails and hull of a yacht are compared with experiment. It is found out that CFD is a very effective tool for the evaluation of sails and hull with keel/rudder in various position of a sailing yacht, which can be costly if they are obtained from towing tank and wind tunnel tests.

**KEY WORDS:** Sailing Yacht; CFD; Turbulent flow; Free surface; Center of effort; Main sail; Jib sail, Keel; Rudder

### INTRODUCTION

Flow analysis around a sailing yacht is much more complicated than a commercial ship, because propulsive forces come from sails in the air and keel/rudder in the water and they should be balanced in yawed and heeled condition. Sails generate side forces as well as propulsive forces since they act like a system of zero-thickness foils with attack angle. The so-called lateral resistance of hull, keel, and rudder below the free surface should balance the side forces from the sails. Otherwise, the yacht would be drifted. The yacht should advance with the leeway angle (heading angle of the yacht centerline) to give attack angle so that the keel and rudder could generate compensating side forces. Both side forces from sails and keel/rudder also act like a coupling moment to heel the ship. The yacht would be heeled until the up-righting moment due to buoyancy will meet the heeling moment from sails and keel/rudder.

The yacht design is usually based upon empirical formula and designer's experience. Towing tank and wind tunnel tests are occasionally utilized. However, it is very costly to get hydro/aero-dynamic information happening around sails and hull with appendages, since there are too many flow parameters concerning yacht's performance, such as wind direction, advancing angle, interference between sails, heeling and yawing angles, although the geometries of

sails and hull are already determined. Computational fluid dynamics (hereafter, CFD) that gains acknowledgement as an efficient tool to assess the ship's performance can be a good candidate for the yacht designer to have enough information a prior to deciding sails and hull with appendages.

In the present study, a 30 feet yacht (KORDY 30), designed and tested by KRISO (presently KORDI/MOERI), is chosen, since the experimental data and computational results are available (Yoo et al., 2005a, 2005b, Yoo and Ahn, 2005, Lee et al., 2006, Yoo and Kim, 2006). In the first, turbulent flows around a yacht hull with keel and rudder were simulated with and without free surface elevation. In the next, flows around main and jib sails with mast were calculated. The Reynolds-averaged Navier-Stokes equations were solved. The calculated results are compared with experimental data.

It is found that the calculation around the hull with appendages including free surface gives better agreement with measurement, and heeling angle affects a great deal on hydrodynamic performance of a yacht. Turbulent flows around a main sail are much influenced by jib sail angle because main sail is located inside of the wake field of jib sail. The calculated center of effort, that is pressure center of the sail system is much different from geometrical center of sail surface, which is commonly accepted as an empirical center of effort.

### GEOMETRY OF HULL AND SAILS OF TEST YACHT

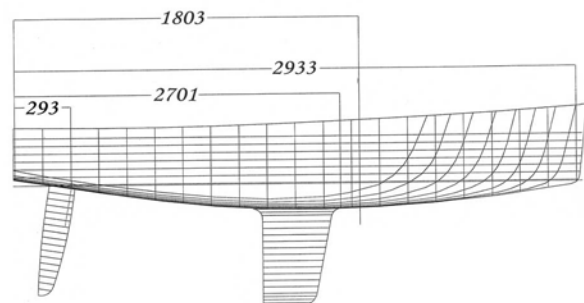


Fig. 1 Shape of a 30 ft yacht hull (KORDY30, model scale)