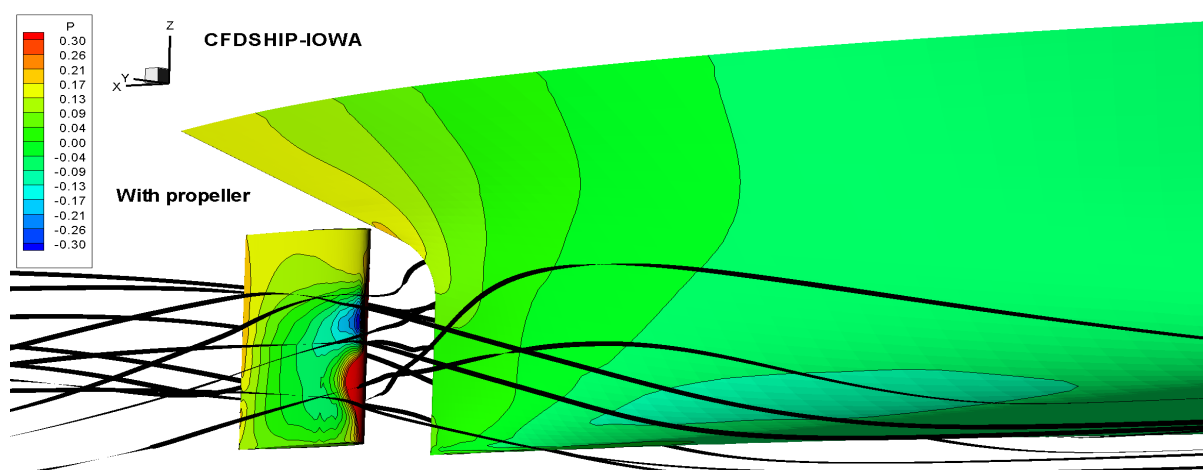


Viscous 3D Flow Simulation



Pressure distribution on the hull and rudder of Esso Osaka, while the propeller is working - Calculated with CFDSHIP-IOWA

Introduction

Successful design and optimization of marine structures requires information about the flow problem on both integral and field quantity levels. Therefore, good insight in the flow problem is essential. Integral data, i.e. hydrodynamic forces and moments, is relatively easily measured in the towing tank, but when it comes to the field quantities, i.e. velocity, pressure and wave fields, extensive data sets are difficult and expensive to obtain experimentally. In order to provide a supplement to the experiment, FORCE Technology is now applying viscous Computational Fluid Dynamics (CFD) for computation of detailed information about the flow problems, which for instance can be used in the design phase.

Computational procedure and numerical tools

A CFD job involves three different activities in which different software packages are applied. The three activities, which are described below, cover: Pre-processing, Flow simulation and Post processing.

Preprocessing

In the preprocessing, the flow domain is discretized, i.e. the domain around the structure is divided into a (large) number of small control volumes, which are required by the flow solver in order to solve the

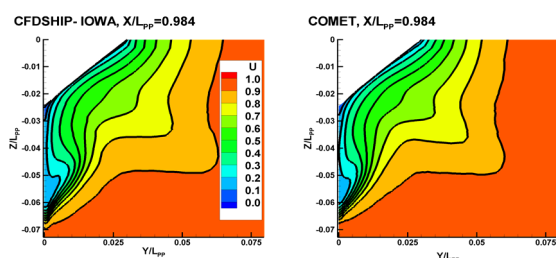
governing equations. The discretization is performed with Icem or Gridgen, which based on a numerical surface representation of the considered geometry, allows the user to generate the computational grid around the structure. At present mainly hexahedral cells are applied, since they give a good resolution of the flow in the boundary layer on the structure.

Flow simulation

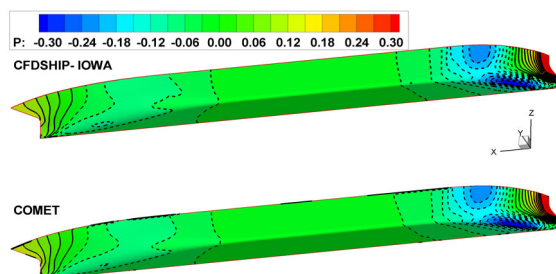
Flow simulations are at present conducted with two different RANS codes: COMET and CFDSHIP-IOWA. The codes solve flow problems including features like free surface and viscosity plus turbulence introduced through different turbulence models. The procedure for running the simulation is to import the grid and set up the solver, i.e. specify 1) appropriate boundary conditions, 2) discretization schemes, 3) material properties and 4) turbulence model. The codes can calculate both steady and unsteady solutions, but the choice of scheme depends on the nature of the considered flow and the intended application of the results. The typical output from a flow simulation consists of a set of field quantities: Three velocity components, pressure, wall friction, free surface elevation and turbulent kinetic energy plus a set of hydrodynamic forces and moments acting on the structure.

Post processing

In the post processing the desired flow information is extracted from the data set generated by the CFD-code. Usually, the data set is large, so flow visualization is important in order to study and analyze the complex flow problems. For this purpose FORCE Technology applies Tecplot, which for instance is used to visualize the flow by means of stream-trace plots, contour plots and vector maps.



Wake field calculation. ESSO OSAKA



Calculated pressure distribution on hull. ESSO OSAKA

Applications

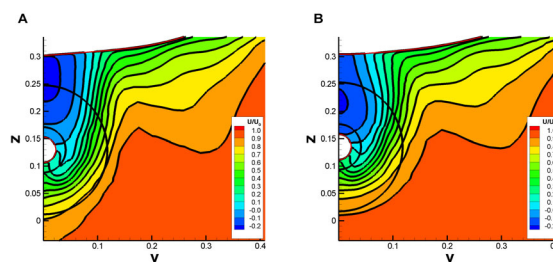
Today calculations are carried out on several vessel types where more detailed flow studies are required. The assignments include alignment and optimization of skegs on twin skeg vessels; wake surveys on full form vessels; appendage alignment; rudder-propeller-hull interaction; etc. Furthermore, CFD is applied in connection with offshore tests, where they are used for general flow field checks in order to guide the design and optimization of the model test set-up.

Case study - variation of hull design

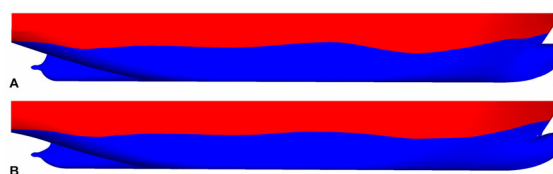
An example of application of viscous CFD is in connection with evaluation and ranking of different hull designs in the early design stage. The results in the figures originate from a case study, where the initial design A of a tanker is compared with an altered

design B, which is modified in the stern and in the fore-body, to improve the wake-field and reduce the total resistance. Concerning the wake-field there is not much difference between the two models, because the modification is relatively small. However, the modified version B appears to give a slightly better nominal propeller inflow than A. With respect to the resistance, it is seen that the difference found in model test, is also found with good accuracy in the calculations. The reduction is also reflected in the wave profiles along the hulls, which indicate that the modified model B gives less wave elevations than A. Finally, the last figure illustrates the streamline patterns on the two versions of the hull. This information is normally found in the streamline test in the towing tank, but with the viscous CFD methods it is possible to get an idea about the flow pattern without testing.

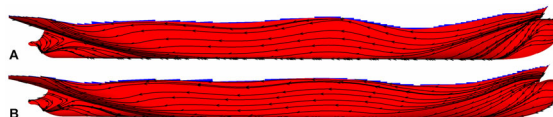
	Reduction in resistance between A and B
Model test	13.7 %
Calculation	12.3 %



Calculated wake fields for hull A and B



Calculated wave profiles for hull A and B



Calculated limiting streamlines for hull A and B



Further information:

Christian Schack, tel. (direct) +45 72 15 78 05, crs@force.dk
 Claus D. Simonsen, tel. (direct) + 45 72 15 77 38, cds@force.dk
 Division for Maritime Industry, Kgs. Lyngby, Denmark, tel. +45 72 15 77 00, fax +45 72 15 77 01

Subject to changes without notice

FORCE Technology USA Inc.
 Tel. +1 713 975 8300
 FORCE Technology Canada Inc.
 Tel. +1 403 286 0606
 FORCE Technology Brazil Ltda.
 Tel. +55 21 2610 7400
 FORCE Technology Netherlands B.V.
 Tel. +31 71 523 5212

FORCE Technology Norway AS
 Claude Monets allé 5
 1338 Sandvika, Norway
 Tel. +47 64 00 35 00
 Fax +47 64 00 35 01
 info@forcetechnology.no
 www.forcetechnology.no

FORCE Technology Sweden AB
 Tallmätargatan 7
 721 34 Västerås, Sweden
 Tel. +46 (0)21 490 3000
 Fax +46 (0)21 490 3001
 info@forcetechnology.se
 www.forcetechnology.se

FORCE Technology
 Main office
 Park Allé 345
 2605 Brøndby, Denmark
 Tel. +45 43 26 70 00
 Fax +45 43 26 70 11
 force@force.dk
 www.force.dk