Validation of CFD simulations of air-lubrication behind interceptor

Anders Östman[1], Fengjian Jiang[1], Kourosh Koushan[1], Luca Savio[1], Pierre-Yves Henry[1]

[1] Ship and Ocean Structures, SINTEC Ocean AS, Trondheim, Norway NO-7052

INTRODUCTION

To understand the performance in terms of resistance reduction for ALS system, it is important to conduct experimental tests and/or numerical simulations of the system. There are several different numerical models which can be chosen to model the two-phase flow, from rather simplistic, such as the Volume of Fluid (VOF), to more sophisticated, i.e. Euler-Euler and Lagrangian models. It is also important validate these two-phase numerical models against high quality experimental tests. The computational requirements to conducts the validation studies may be massive.

In the present study a VOF method is utilized, the simulation results are compared against experimental tests conducted in the SINTEF Ocean water tunnel. The ALS system consist of a suction creating interceptor located upstream of the air entry holes. The system is aimed for shallow draft, high speed vessels, where the suction generated by the interceptor is sufficient to suck air that feeds the air layer beneath the hull. The system is currently in use on the MS Fjordøy high-speed catamaran, which is in service along the Norwegian west coast.

CFD VALIDATION STUDY

A quasi-2D approach was chosen for the computational domain, this to simplify the CFD simulations, aiming for as similar flow conditions as in the experiments as possible, without modelling the whole 3D water tunnel geometry. The SINTEF Ocean water tunnel consist of a test section with rectangular cross-section. Upstream of the test section there is a nozzle and downstream a diffusor. Both the inlet of the nozzle, and the outlet of the diffusor has circular cross-sections, see Figure 1. To achieve the same cross-section averaged momentum distribution in the CFD simulations as in the water tunnel experiments, the quasi-2D domain is constructed with a non-constant width. The quasi-2D width is determined based on the cross-section area variation of the water tunnel.

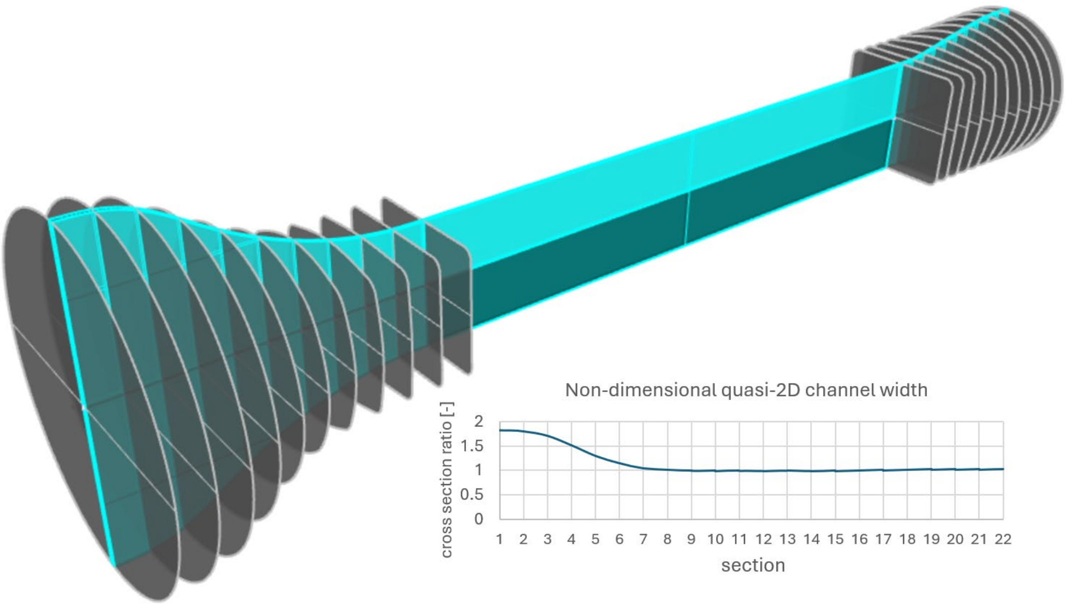

Figure 1 Cross sections of the nozzle, test section and diffusor of the SINTEF Ocean water tunnel. The quasi-2D domain used in the CFD simulations is visualized by the cyan coloured object.

The StarCCM+ RANS flow solver was chosen for the simulations. The two-phase flow was modelled using a VOF technique applying the HRIC scheme. Turbulence was modelled using the k-ω SST two-equation turbulence model. Results from the CFD simulations will be presented and compared against experiments for water speeds ranging from 4 to 7 m/s at four different air injection flow rates.

Acknowledgement: The research was performed under the project OnAir (317869) supported by the Norwegian Research Council and industrial partners Jotun, GS Marine Produksjon, Rødne, Norled, Kumera and Servogear.