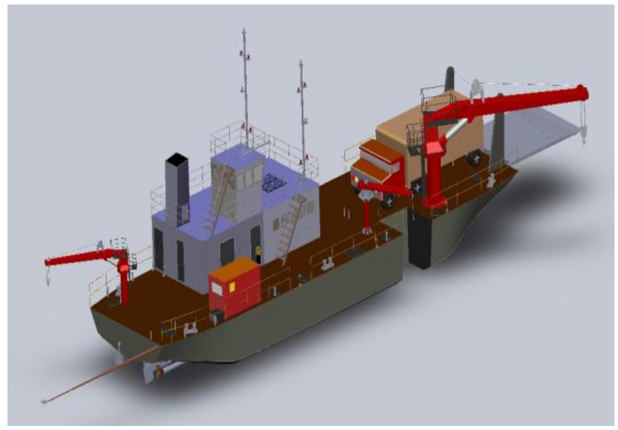
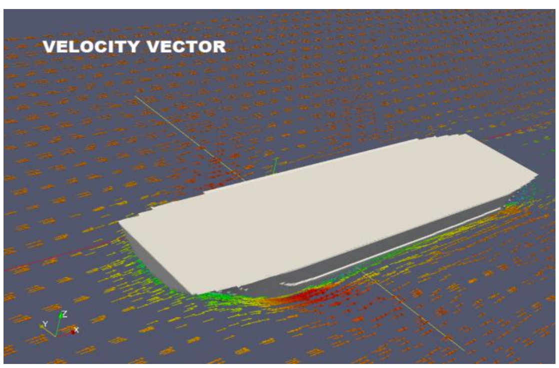
RESISTANCE CALCULATIONS FOR RO RO BARGE USING CFD

BACKGROUND

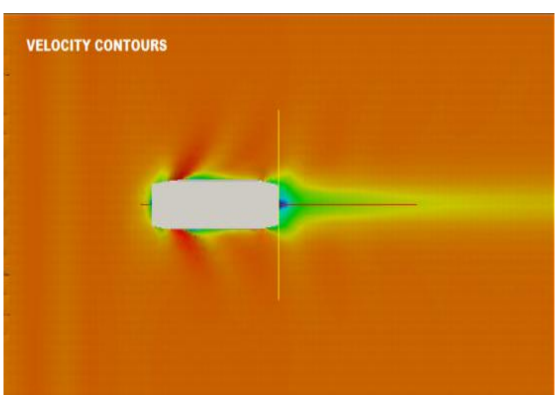
Design of an IR class Ro Ro Barge for the Indian Defense was one of the prestigious project which we undertook. One of the challenging task in this project was to understand the resistance and powering for the vessel. As an experienced design house we know that correct powering prediction is the first step in a successful vessel design. Hence we used CFD analysis for resistance calculations barge and compared the results with the experimental model tests. Details of one of the iterations were as follows.





ANALYSIS – PROCESS

The 3D model of the barge was prepared and imported into CFD software. 3D domain was created around the hull form. The complete domain was meshed, with a refined mesh around the hull for better resolution of boundary layer flow. The boundary layer thickness and number of layers were adjusted to account for turbulent flow around the barge hull.

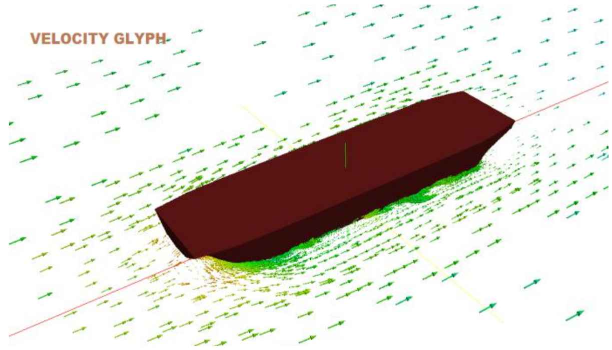


Multi-phase analysis was done in this study by using the interface-capturing approach, which solves the RANSE equations on a predetermined grid that covers the entire domain, and the Volume of Fluid (VOF) method, to simulate the free surface flow for the barge. For turbulence modeling, the Shear Stress Transport K – ω (SSTK - ω) model was used for the numerical solution.

The boundary condition specified was similar to that of the experimental setup. The motion of the free surface was governed by the gravitational acceleration. The CFD domain was thus modeled as open channel flow, consistent with the experimental setup.

At the inlet, a uniform flow was specified. The hydrostatic pressure at the outlet was calculated assuming an undisturbed free surface. Smooth walls with a no slip condition were assumed for the top, floor and the sidewall.

Finally a multi-phase transient solution was carried out to calculate the resistance on the barge. The result plots show the wave pattern, water elevation, velocity streamlines, velocity vectors and pressure on the hull.



WATER SURFACE

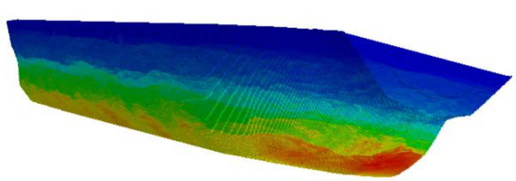
MODEL TEST

Model test was conducted to validate the CFD results :

Vessel had a length of 25 meter , a depth of 7 meter and have a design speed of 9 knots . To suit the model testing facility a scaled down model (1:9) was considered for hydrodynamic calculations. Model was ballasted to represent the loaded condition.

Results were obtained by extrapolation on the basis of ITTC 78 prediction method using froudes law of scaling.

HULL PRESSURE CONTOURS



RESULTS

The results of CFD analysis was a good match with the experimental study conducted later. The CFD method, in comparison with the model test, has great advantages, such as lower costs, reduced time, accessibility and more visible details in results e.g. pressure and velocity contours, vectors and gradients. It also allows designers to calculate the hull resistance and investigate the flow around the hull without having to do a towing tank test.