KCS Bare Hull Resistance

Dr. Joseph Prabhu & Mr. Ram Kumar Joga
Research and Development,
Indian Register of shipping
H.O, Mumbai

Synopsis

In this research migration project, the bare hull resistance of Kriso Container Ship (KCS) is computed in OpenFOAM-9. The resistance computation involves a multiphase flow with two phases air-water are captured using Volume of Fluid method. As the flow physics operates in high Reynolds number two equation turbulence models are used to resolve the turbulence.. In this project, the results of ship resistance are estimated using *k-Omega-SST* turbulence model is used with adaptive time step.

The computational domain is meshed using *blockMesh*. Mesh refinements are provided at various locations of air-water interface, bow, kelvin-wake and in the ship wake region. The refinement blocks were prepared using *toposet* module of openFOAM with five levels of refinement. The cleaned KCS hull geometry is imported in the '.stl' format and final meshing is carried out using *SnappyHexMesh*. The computational domain is modeled similar to the towing tank. Since, the hull is symmetry only half the domain is solved by giving mid-plane symmetry boundary condition.

The simulation is performed in OpenFOAM-9 with *interFoam* solver. The run is carried out in parallel using 48 core server which took 3-6 hrs to complete the simulation of 73-190 sec for speed ranging from 2.38 m/s to 0.912 m/s with a time step of 0.0001 sec. The X-component drag component gives the resistances of the vessel and it is seen that there is a variation of maximum 9% in higher Froude numbers.

References

1. Marcu, O., Obreja D., "Model tests on the KRISO hull for the powering performance assessment", The Annals of "Dunarea deJos" University of Galati, Fascicle XI-Shipbuilding, pp. 17-22, 2011.

1. Introduction

The bare hull resistance of Kriso Container Ship (KCS) is computed in OpenFOAM and validated with model test and CFD (StarCCM+). The model test is carried out by Korean Research Institute for Ships and Ocean Engineering (KRISO). The towing tank experiments were carried out by KRISO to obtain the resistance, mean flow data and free surface waves. The dimensions of the towing tank is of length of 200 m and breadth of 16 m and a depth of 7 m. The model is prepared in the scale of 1:31.6 of length 7.27 m.

2. Governing Equations and Models

The governing equation of the fluid flow can be written as Eq. 1

$$\frac{\partial}{\partial t}\int_V \rho dV + \oint_A \rho \, \mathrm{v}.da = \int_V S_u dV \qquad (1)$$

The flow momentum equation is written as Eq. 2

$$\frac{\partial}{\partial t}\int_V \rho dV + \oint_A \rho \, V \otimes V.da = \oint_A pI.da + \oint_A T.da + \int_V f_b dV + \int_V S_u dV \qquad (2)$$

Where,

t	-	Time
V	-	Volume
a	-	Area vector
ρ	-	density
v	-	velocity
S_u	-	Source term
p	-	Pressure
T	-	Stress tensor
f_b	-	Body forces
S_u	-	Source term

To solve the momentum equations governing the velocity field of a fluid, it is necessary to establish a connection between the stress tensor and the velocity field. This connection is known as closure and requires the use of various constitutive equations that take into account the fluid's material properties, including viscosity, first and second normal stress coefficients. Additionally, the closure of the system of equations described above requires the inclusion of equations of state, which are also considered as constitutive relations.

3. Simulation Procedure

1. Geometry and Mesh

The principal dimensions of the KCS model is given in Table 1 and the CAD model used for the CFD run is shown in Figure 1.

Table 1: Principal dimensions of KCS model hull

Particulars	Dimensions
Length between perpendiculars (LBP)	7.27 m
Length of waterline (LWL)	7.3570 m
Breadth of Waterline (BWL)	1.0190 m
Depth	0.6013 m
Draft	0.3418 m
Displacement	1.6490 m^3

Figure 1: CAD model of KCS hull

The computational domain is prepared as per ITTC 2014 guidelines and the details are given in Table 2 and can be visualized in Figure 2.

Table 2: CFD computational domain

Computational Domain	
Front	2.5 L
Aft	2.5 L
Side	2.0 L
Bottom	2.0 L
Top	1.0 L

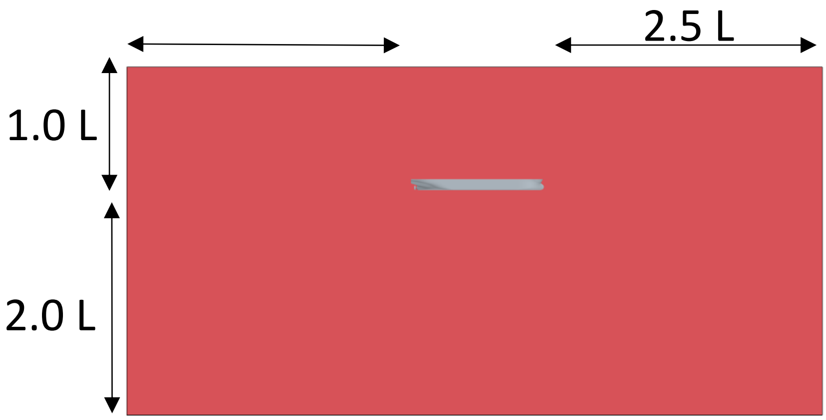

Figure 2: Computational domain used for simulation

The computational domain is prepared using BlockMesh with waterline interface refinement. The refinement zones in the mesh are prepared in various levels with the help of topoSet file as shown in Figure 3. After creating the refinement zones, SnappyHexMesh is used to get the final mesh for the simulation as shown in Figure 4.

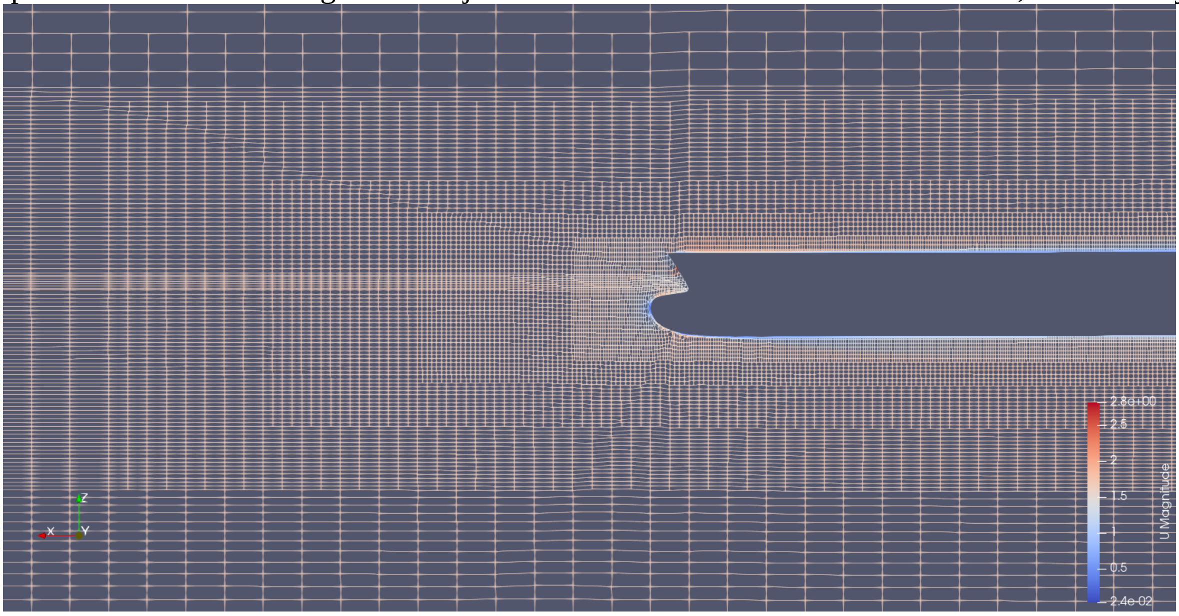

Figure 3: Refinement zones in the computational domain

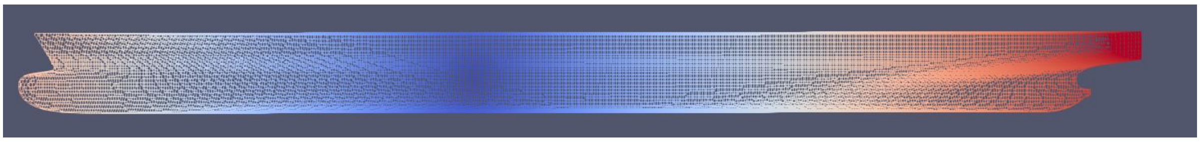

Figure 4: Meshed hull using SnappyHexMesh

2. Initial and Boundary Conditions

The boundary condition is considered as given in Table 3. The simulation is carried out for 4 different speeds of 0.912, 1.646, 1.91 and 2.38 m/s accordingly the Inlet, Side, Bottom and Top boundary conditions needs to be changed.

Table 3: Boundary conditions applied in computational domain

Boundary Conditions	
Inlet	Velocity
Outlet	Pressure
Side	Velocity
Bottom	Velocity
Top	Velocity

3. Solver

The solver used is ‘interFoam’ and the simulation is run parallel in 48 cores. The turbulence model ‘kOmegaSST’ is used. As the hull is symmetry only half of the domain is considered in simulation to reduce the computation expense. The computation domain is cut in the mid-half and boundary condition of symmetry is applied in that plane. The time interval is taken as 0.0001 sec and simulation is run for 50 sec. The solution is converged after 30 sec.

4. Results and Discussions

The resistance is computed for various Froude number as tabulated in Table 4. The solution is converged after 30 sec and the resistance for Froude number 0.195 is 43.36 N as shown in Figure 5.

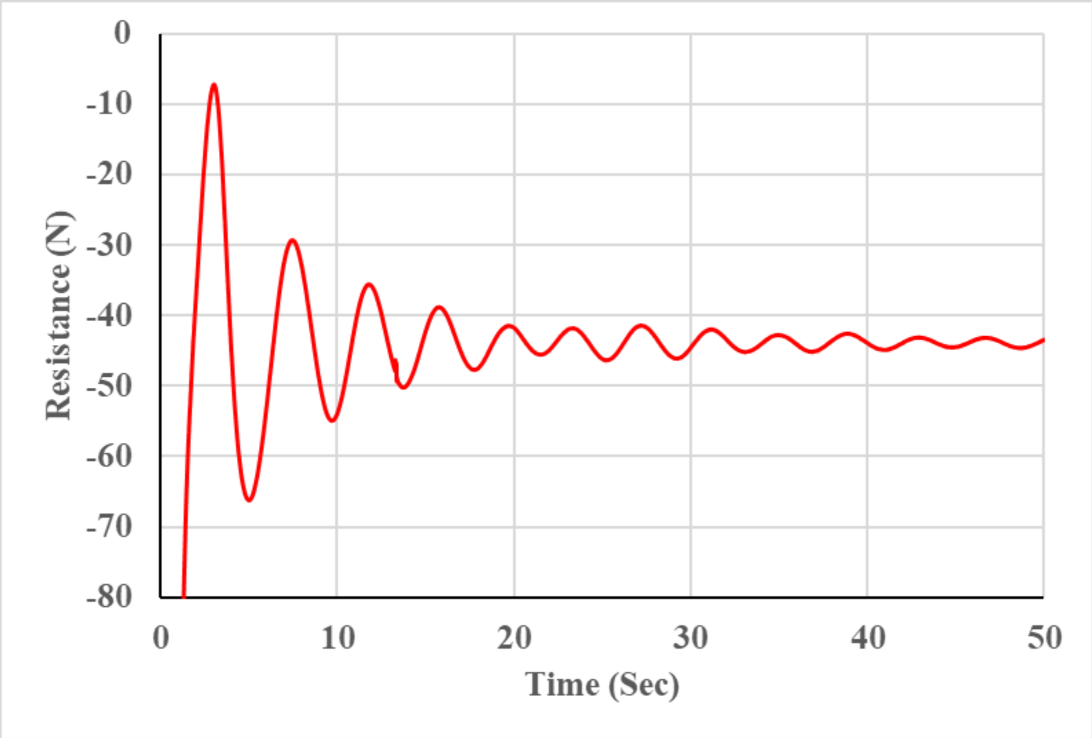

Figure 5: Resistance convergence of Froude number 0.195

Table 4: Comparison of StarCCM results with OpenFOAM

Fr	Velocity	StarCCm			OpenFoam			% Varaiation
		Rp(N)	RF(N)	RT(N)	Rp(N)	RF(N)	RT(N)	
0.108	0.912	1.7	12.86	14.56	2.94	12.68	15.62	-7.28
0.195	1.646	5.22	38.75	43.97	6.48	36.88	43.36	1.39
0.227	1.91	8.08	51.82	59.9	13.9	50.2	64.1	-7.01
0.282	2.38	41.07	79.03	120.1	42.58	73.8	116.38	3.10

Table 5: Comparison of experiment results with OpenFOAM

Fr	Velocity	Experiment	OpenFoam			% Varaiation
		RT(N)	Rp(N)	RF(N)	RT(N)	
0.108	0.912	15.4	2.94	12.68	15.62	-1.43
0.195	1.646	46.7	6.48	36.88	43.36	7.15
0.227	1.91	63.2	13.9	50.2	64.1	-1.42
0.282	2.38	128.9	42.58	73.8	116.38	9.71

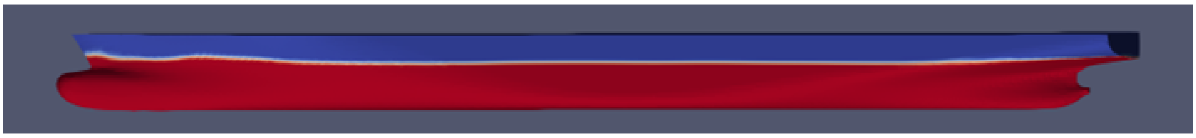

Figure 6: Free surface effect along the hull surface

The computed results from OpenFOAM is compared with StarCCM and experiment data and is tabulated in Table 4 and Table 5 respectively. It is noted that there is variation of maximum 9 % from experiment and StarCCM for higher Froude numbers. This variation can be minimized in further refinement in meshing and that will be done in future developments.

5. References

- ITTC 7.5-03-02-03:2014. Recommended procedures and guidelines. Practical guidelines for ship CFD applications, revision 01.
- Marcu, O., Obreja D., “Model tests on the KRISO hull for the powering performance assessment”, The Annals of “Dunarea deJos” University of Galati, Fascicle XI-Shipbuilding, pp. 17-22, 2011.

DISCLAIMER: This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.