IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X,
PP 68-74
www.iosrjournals.org

Numerical Modelling For Drag of a Small Water-Plane Area Twin Hull Vessel with Experimental Validation

Momin Azaruddin S. [1], Dr. V. Anantha Subramanian [2]
[1] *(Department of Naval Architecture & Offshore Engineering, AMET University, India)*
[2] *(Department of Ocean Engineering, IIT Madras, India)*

Abstract: *Small water-plane area twin hull (SWATH) vessels have distinct geometric features to give them sea-friendliness characteristics. The assessment of the basic characteristics through re-design and assessment are important evolutionary investigations to obtain a good hydrodynamic design. A case study is presented here using a commercial code namely, STAR CCM+ to simulate the steady speed and obtain the kinematics as well as the drag. A study of this kind has value only when necessary validation is performed. For this purpose a scale model has been chosen and towing tank tests have been conducted to independently obtain the flow wave pattern and also the very important characteristic of drag through physical measurement. The computationally obtained values are compared with experimentally measured drag values and presented here. The paper therefore establishes the validation of a numerical approach along with details of the numerical modelling scheme. An additional important characteristic of the SWATH form is that because of the low exciting forces associated with heave and pitch, these motions can be controlled by the use of active fins and to a significant extent by the use of passive fins.*

Keywords: *CFD, Drag, Flow kinematics, RANS, SWATH*

I. Introduction

The standard mono-hull ship is very old concept and some deficiencies have been noticed and confirmed in terms of stability of ship in rough sea condition. One of the major problems is inability to operate on rough seas with high performance or even inability, for certain designs, to operate at all. Modern demands on naval ships require that they should have broad capabilities, not only for survey and war operations, but also for control of the sea transport, repression of goods as well as other capabilities like participation in rescue missions and pollution prevention. These requirements condition some new design solutions for ability to perform various duties on rough seas as well as larger deck space for the installation of various equipment necessary for diverse operations and/or even modular deck parts.

The multihull concept has been suggested as a potential solution. The SWATH concept has been proposed as a solution, Small Water-plane Area Twin Hull (SWATH) ship originated as an inspiration from the form of semi-submersibles with their characteristic submerged pontoon hulls and prismatic support legs with relatively small water plane area. SWATHs have been in existence for more than the last three decades. It consists of a pair of lower hulls, which are connected to the upper platform by single or twin strut. It has small water-plane at a designed draft, which results in longer natural periods and reduced buoyancy force changes. As a result of the geometry, SWATH ships have a remarkably stable platform configuration which is highly attractive for science and engineering operations at sea. The SWATH vessel has some disadvantages like, reduced damping and restoring forces in pitch and heave. Hence introduction of passive and active control devices can help to control the pitch and heave amplitude, rate and acceleration of the motion as well as some of the associated dynamic effects .Any pay-offs due to the introduction of such devices must also be analysed. Possible effects are like increased drag leading to extra power requirement, induced wave interaction effects between the two hulls etc.

II. Geometry and test condition

Simulations were carried out on the SWATH 10 vessel [4]; the geometric form and fin data are given in Fig. 1 and summarized in Table1.In the first simulation, the model is free in all degrees of freedom in forward speed condition in the Froude number range 0.12 to 0.36 for the bare hull. In the next simulation, augmentation of drag is analyzed due to presence of fins for the Fn = 0.25, when ship is free in all degrees of freedom for with and without canard fin and aft fin. For simulating stabilizer fin action, the aft fin angle is fixed at -15^{0}.

Fig.1: Hull form and main geometric data of SWATH10

Table 1: Main particulars of SWATH 10 model

Specification	Model (1:24)
Displacement (T)	0.043
Strut length (m)	1.587
Hull length (m)	1.573
Maximum beam (m)	0.75
Strut + hull wetted area (m^2)	1.493
VCG above the baseline (m)	0.241
LCG aft of strut nose (m)	0.6568
GM_L(m)	0.3105
Pitch radius of gyration (m)	0.4764
Draft (m)	0.184
c_b	0.216
Canard fin LE distance from nose of Hull	0.16
Stabilizer fin LE distance from nose of Hull	0.95

III. Computational and experimental analysis

3.1 Computational analysis

Numerical solution of any fluid dynamics problem involves solution of conservation of equations of fluid motion, namely the mass and momentum conservation equations. The momentum conservation equations, also known as Navier-Stokes equations, lead to RANS equations by time averaging procedure. For an unsteady incompressible flow, the continuity and Navier-Stokes equations can be written as:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = \rho g_i - \frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial {x_j}^2} \tag{2}$$

For turbulent flow, the instantaneous velocity field u_i is decomposed into the mean and fluctuating components as $u_i = U_i + u'_i$. Using this decomposition in (1) and (2) and carrying out the time averaging procedure, one obtains:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{3}$$

$$\rho \frac{\partial U_i}{\partial t} + \rho u_j \frac{\partial U_i U_j}{\partial x_j} = \rho g_i - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \mu \frac{\partial^2 U_i}{\partial {x_j}^2} \tag{4}$$

The realizable k-ε model has been used in the present work as it has been verified in literature for accuracy and robustness.

3.2 Body equation of motion

The motion of rigid body is governed by the conservation linear and angular momentum which is given as:

$$\sum F_{b_i} = m \frac{d^2 x_{b_i}}{dt^2}$$
$$\sum M_{b_i} = \frac{d}{dt}\left(I_{ij} \frac{d\theta_{b_j}}{dt} \right) \tag{5}$$

3.3 Domain, grid and boundary conditions

The computational domain around the hull is shown in Fig. 2 where velocity inlet is at 1L from bow, pressure outlet is at 5L from stern, exterior boundaries are at 1.5 L from starboard and port sidewall of hull and the bottom boundary is at 1.5L from the base line keel of hull. Here, hull x - axis is aligned to x - axis of earth coordinate system.

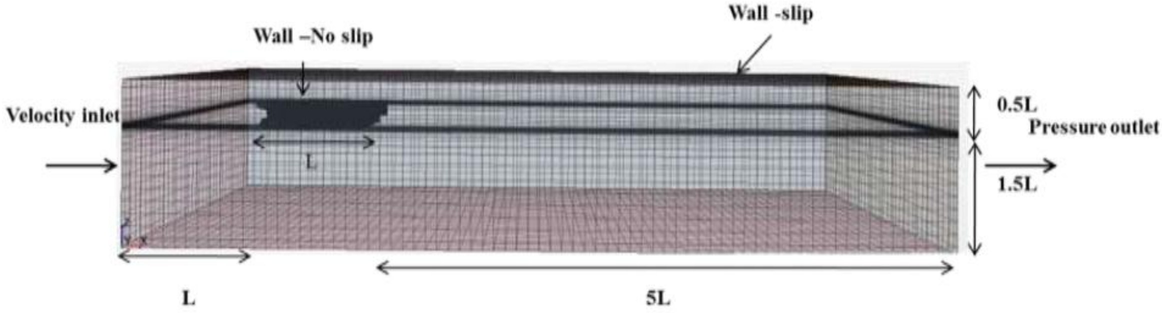

Fig.2: Domain detail

The boundary conditions used for computational analysis are mentioned below in Table 2

Table 2: Boundary conditions

Boundary type	
Inlet boundary	Velocity inlet
Outlet boundary	Pressure outlet
Exterior boundaries	Wall – slip
Hull	Wall - No slip

The domain is divided into three regions for grid generation, these regions differ in terms of the size of the cells and the associated growth ratio of the cell heights perpendicular to the surface of interest. The meshing uses unstructured grid created by means of mesh generation tools of Star CCM+. The fluid volume in the computational domain surrounding the hull is discretized by employing a hybrid grid approach. Predominantly hexahedral cells are used; polyhedral cells are used next to the hull surfaces to best capture their curvature. At the free surface, the volume grid is clustered to capture the high flow gradients and free surface deformation prism layers are provided to capture flow properties near boundary of the vessel. The volume mesh or grids around the hull and volume mesh on hull are shown in Fig.3 and 4

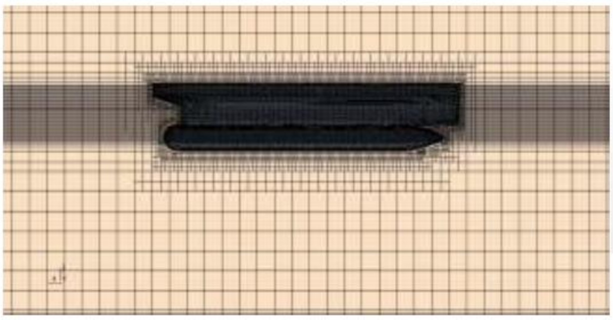

Fig.3: Volume mesh around the hull and at free surface

Fig.4: Volume mesh on the hull

Grid independence study has been performed to give stability of results for the smaller use of cells used. Different parameters are picked obtained from CFD and data were also compared to perform the grid independent study. Stability of results was obtained with the use of 2.1 million cells and also with comparisons with the experiments for bare hull and 2.5 million for hull with fins. The comparative values of the bare hull shown in Fig.5

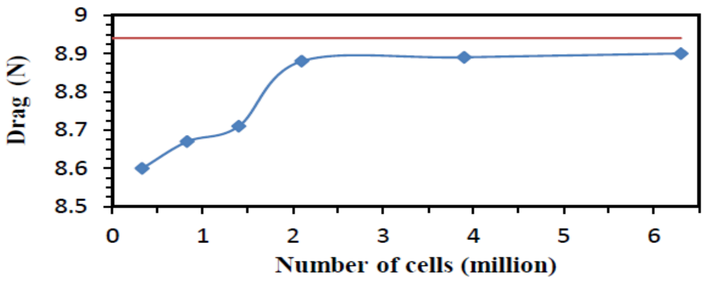

Fig.5: Grid independent study for bare hull

IV. Experimental setup for drag analysis

The numerical modelling and simulation involve the interaction of canard fin and aft fin respectively with the hull form geometry. It is necessary to validate the values of hull Drag prediction in forward speed condition in calm water condition. To complete the validation, experimental studies have been carried out Towing tests to obtain Drag and flow kinematics. The model for the physical test is prepared, and the Drag tests are performed according to Froude law of scaling at the Towing Tank facility, Indian Institute of Technology, Madras according to the International Towing Tank Conference (ITTC) recommendations. The procedure followed is the ITTC 78 prediction method. The tank dimensions are 82.0m × 3.2m × 2.8m (water depth) and the maximum towing carriage speed is 4m/s. The towing carriage is equipped with fully automatic speed control using closed-loop feedback control system for the synchronized drive motors of its four wheels.

4.1 Model preparation

The FRP hull of the vessel model is manufactured to the geometric scale according to the ITTC recommended procedures. The model is marked with the frame number and water lines as per the lines plan drawings. The ballast weights are distributed inside the hull to achieve even keel floating position, as is the case, or to the required trim condition, without heel by visually inspecting and ensuring that the waterline of the floating model matches with the corresponding draft line drawn on the hull. A tow point is fixed to the base inside the hull at approximately mid-ship location, on the centre line, to connect the towrope that transmits the hull drag force to the load cell through a pair of pulleys.

4.2 Instrumentation

The dynamic force of the towed model is measured using a beam type load cell. The load cell used is of nominal rated load 50kg (≈500N of force), it is calibrated for the measurement range, and the data is keyed into software (Catman). The calibration curve for the load cell is given in Fig. 6.

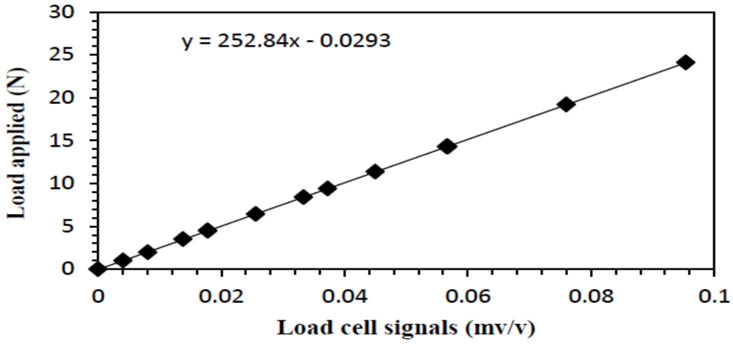

Fig.6: Load cell calibration

4.3 Drag test procedure

The model is fixed to the towing carriage and held with a brake in the condition while the carriage is at rest, accelerating phase or reversing after a run. The model is connected to the carriage and towed in such a way that during the test, when the brake holding the model is released, it is free to take its natural trim and heave attitude while the carriage tows the model along the centreline without any drift or sway. The recorded data of the measured force for each run is processed to obtain the average tow force value by selecting the data window for the constant speed, duration of the time series plot corresponding to the particular speed. Fig. 7 shows an experimental facility and setup for Drag test. The experiments are performed for the forward speed ranges for Fn = 0.12 to 0.36.

Fig.7: Experimental setup for drag test

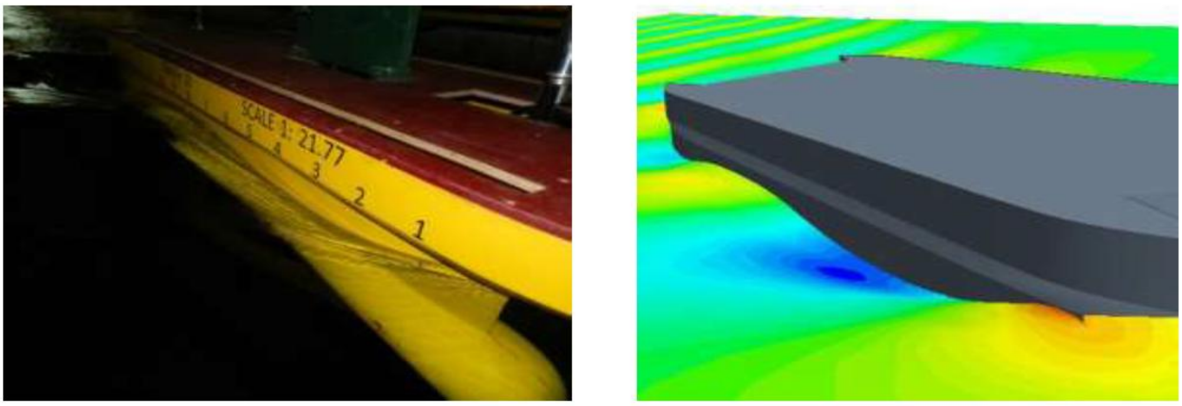

Fig.8: Flow pattern freeze shot around the hull from experiment and numerical simulation result from CFD. Flow kinematics are captured very well in the numerical model.

V. Results and discussion

Fig. 8 shows the flow pattern around hull during the run when the model is running for 1.45 m/s (Fn = 0.36). Flow pattern obtained from experiment shows matching reproduction of the wave pattern around the hull in the numerical simulation. It is also noteworthy that the pattern of total Drag coefficient is very well captured with a rising peak at Fn = 0.31 and a subsequent reduction. The magnitudes do not match exactly however; the trends are captured very well. See Fig. 9.

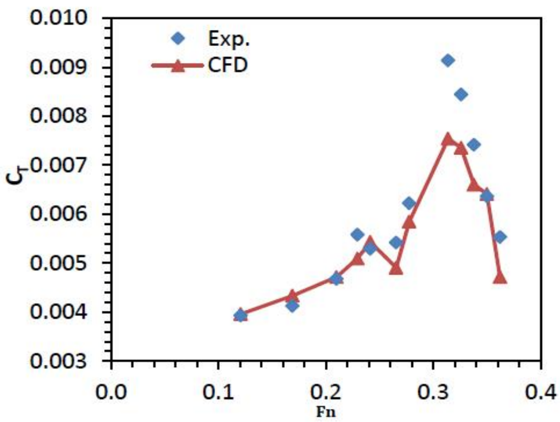

Fig.9: Drag coefficient of un-appended hull for different speed

Fig.10: Experimental setup with fins attached to hull

Fig. 10 shows the experimental setup for drag test of model with fins attached to it. Canard fin angle was set to 0^0, 10^0 and 20^0 and experiments were conducted to obtain the Drag values. These were compared with results from numerical simulations. For Fn = 0.25, the results on model scale are directly compared in Fig. 11 as non-dimensional drag coefficient. The fins obviously contribute significantly to the drag of the vessel.

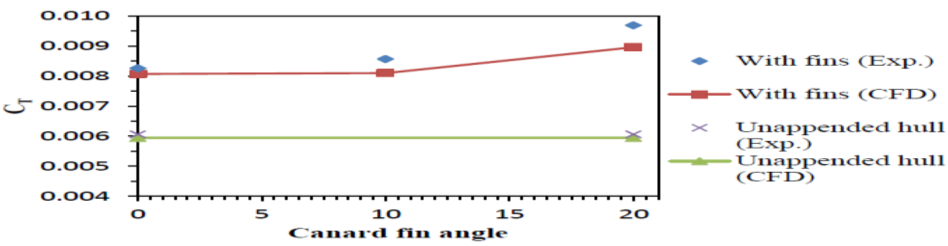

Fig.11: Drag coefficient of hull for different canard fin angle (Fn = 0.25)

VI. Conclusion

The study validates the application of numerical method in predicting the drag as well as kinematic aspects of a SWATH vessel. The study has been extended to quantify the influence of canard and aft fins which are useful in bringing down the motion characteristics in waves. The increase in drag due to the fins has been quantified and presented. The numerical approach is very useful to establish the drag characteristics in the initial design stage.

References

[1] Ochi, K. M., "Hydro-elastic Study of a Ship Equipped with an Anti-pitching Fin," *published as DTMB Report* 1455, 1962.
[2] Azcuta, R. Ship Drag prediction by free-surface RANS computations. Ship Technology Research, 47(2), 2000, 47-62.
[3] S. Kos,D. Brcic,and V. Francic, “Comparative analysis of conventional and SWATH passenger catamaran”, research report, University of Rijeka, 2009.
[4] Lewandowski,E.M., "Experimental Study Of An Automatic Pitch Control System On A Swath Model"*Davidson Laboratory Technical ReportCG-D-12-94*, 1994.
[5] F. Granneman, “SWATH – A proven and innovative hull concept”, Abeking &Rasmussen Schiffsund Yachtwerk Aktiengesellschaft, 2011.