2021 International Bhurban Conference on Applied Sciences and Technologies (IBCAST) | 978-1-6654-0516-4/21/$31.00 ©2021 IEEE | DOI: 10.1109/IBCAST51254.2021.9393279

RESISTANCE PREDICTION & RUNNING ATTITUDE OF HIGH-SPEED SEMI-PLANING HULL USING COMPUTATIONAL FLUID DYNAMICS

Syed Muhammad Meesam
National University of Science & Technology
Karachi, Pakistan
engr.meesam@outlook.com

Dr Zeeshan Riaz
National University of Science and Technology (NUST)
Karachi, Pakistan
z.riaz@ucl.ac.uk

Dr Hassan Khalid
Platform Design House
Karachi, Pakistan
hassan.khalid@uclmail.net

Dr Muhammad Saeed Khalid
National University of Science and Technology (NUST)
Karachi, Pakistan
dr.saeed.khalid@gmail.com

Dr Saeed Akram Malik
CESAT, Hydrotechnologies
Pakistan
saeed2k16@gmail.com

Asim Zubair
Platform Design House
Karachi, Pakistan
asimzobair@gmail.com

Abstract— **Precise resistance estimations and running attitude are crucial factors in the design and production phase of hulls. Model research has conventionally relied on experimental model testing, but this methodology is both costly and time consuming. A precise estimation of ship's hull resistance yields the accurate required propulsive power leading to significant cost saving. To estimate the resistance in the design process, Computational Fluid Dynamics (CFD) methods are extensively implemented. Though, even with the maturity of modern day CFD tools, experimental measurements are still the most reliable solution for resistance measurement. However, as said earlier, due involvement of cost effects, CFD can be seen as near accurate replica of experimental facility, provided the results are well validated. CFD simulations for semi-planing hull is an extremely challenging process. However, an effort has been made to address the issues in simple manner. The simulations are carried out to predict resistance in both displacement and planing modes. Numerical Computation is done by solving RANS equations. Multi-phase Volume of Fluid (VOF) method is used for capturing the free surface. For turbulence and kelvin wake pattern K-Ɛ & SST K-ω models are used. Resistance of a semi-Planing hull is also predicted using empirical methods. A comparative analysis between the results from empirical methods, experimental results and CFD results is undertaken to get a validated estimation technique for resistance of Semi-planing hull. CFD simulations has been carried out using ANSYS Fluent software on Semi-displacement R/V Athena hull (DTMB 5365). The CFD results are in good agreement with experimental and empirical results. Once resistance prediction technique for CFD was finalized, the simulations were carried out on a Test hull to predict its resistance and running attitude. Recommendations are made to optimize the test hull design by adding lifting rails and spray rails on several positions of hull geometry.**

Keywords— Semi-planing Hull; Displacement Mode; Planing Mode; Empirical formula; Multiphase VOF; CFD

I. INTRODUCTION

Naval Architecture is an engineering discipline established over the course of hundreds of years. Currently, the primary priority is on fulfilling the specifications of contract speed with minimal fuel usage. To be able for selection of an adequate propulsion unit, it is necessary to make an exact estimate of the hull drag. Investigating the hull's running attitude to get better seakeeping properties is also of concern.

Conventionally, the body resistance and running attitude have been established by performing towing tank experiments on models. These tests have been proved to estimate the running attitude of full-scale models very well. But still this strategy is tedious, costly and just appropriate for the model on which it is applied. Computational Fluid Dynamics is a more general method for prediction of hull power. It is computer simulations where the behavior of a fluid flow system is analyzed utilizing numerical methods.

The methods to perform CFD simulation for Displacement hulls provides very efficient results. CFD simulation of Planing and semi-planing craft is a complex process and it needs to have a proper logical approach and numerical solution. As this study targets only Semi-Planing hull, CFD techniques in this regard are still immature. This research will focus on getting a validated technique to estimate resistance of Semi-Planing hull.

A CFD simulation will be performed for semi-planing hull designed for naval use. This study could further be used by any design house in Pakistan to improve the design process of hull design, i.e. time, cost, and fuel.

II. BACKGROUND

Hull construction had been typically dependent on scaled model testing in towing tanks. Instead of doing these tests, empirical techniques supported statistical method of experimental data and these are developed to predict the behavior based on hull shape and its characteristics. Two commonly utilized empirical techniques/models are Holtrop and Mennen [1], this is incorporated for the assessment of displacement hulls while the second one is Savitsky [2], this is used for planing hulls. The main disadvantage of these empirical models is that they are limited to estimate the resistance of ordinary hull shapes which are geo-symmetrical to hull shapes utilized to design the models.

In previous years, A lot of numerical research on planing hulls are performed. Comparison of CFD simulations with Savitsky by Brizzolara and Serra [3]. The fluid flow was simulated with the RANS & consequently the k-ε and SST k-ω model was selected to capture turbulent flow. The Volume of Fluid (VOF) approach was carried out to determine free surface. It came into observation that hull drag computed using CFD simulations deviated within range of 10% of experimental results. It is much less than the error resulted from Savitsky approach, and this shows the ability of CFD simulations.

To find a vessel's running attitude, it must include its motion within the simulation. To this end, methods are built to couple fluid flow and body motions. In 2001, Azcueta [4] introduced a methodology in which the hull-fluid relationship was modelled. Using the RANS equations, the turbulence was modelled and thus the regular k-ε model was captured using a VOF process, with the free surface used. For a Series 60 Hull this technical method has been proven which can be an established displacement hull, incorporated for benchmarking, by comparing the results with the data provided by experimental results. The overall drag was under-predicted by 5.9% and thus the trim and sinkage were 6.0% and 8.2%, respectively, under-predicted by experimental tests.

Recent studies show that CFD displacement hull simulations are simulated with an accuracy that begins to approach the accuracy of towing tank experiments. 33 participating groups conducted simulations of three broad displacement ships at the Numerical Ship Hydrodynamics Workshop in Gothenburg 2010 [5-pg.1–16]. The results from all simulations show that the resistance predictions have a mean error. Only 0.1% with a typical deviation of 2.1% compared to towing tank experiments. The sinkage and trim estimations were less precise, mean errors were around 4% for the high speeds.

CFD of Planing hull simulation is harder than displacement hulls, and typically the predictions are less precise. During the flow around a planing hull the resulting forces directly depend on the hull's wetted surface, and it is directly depended on the location within the water. Therefore, it is critical to estimate how the planing hull conducts in water prior to satisfactory estimates of drag are often made. [6] Nonlinear effects for example wave breaking and sprays are also becoming more common within high speed ranges. However, as the computing power is increasingly growing and better models are being created, the results of numerical Planing hull simulations are constantly improving. [7] Modern studies [8, 9] have indicated that the CFD of planing hull can produce results that deviate from experimental data with an error of less than 10%.

III. METHOD

The strategy is based on the guidelines provided by the software developer. First of all, a computational domain is defined required to run the simulation and mesh is generated in the entire region according to need. In the model definition, appropriate model is chosen and setting of boundary conditions is done. Solution steps are selected according to the requirement. After obtaining the results, they are analyzed in post processing. This is an iterative process because simulations have to be performed on different conditions.

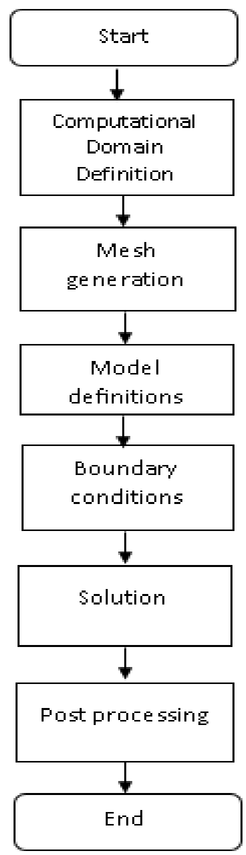


Figure 1: CFD Flow Chart

IV. CFD SIMULATIONS

The approach used to run CFD simulations, including definition of Computational domain, mesh generation, definition of computational models and their properties, setting of boundary conditions, solution and the post processing will be discussed in this section.

A. *Computational domain definition*

To avoid the affects produced by wall boundaries to flow near hull a larger domain was established. To prevent reflections from boundaries of the domain, kelvin wake pattern method was applied to calculate the required length and breadth of the domain. Since only the drag and dynamic lift of the hull is studied so only half of the hull is included along the longitudinal plane of symmetry, In the figure (2) given below, the domain and its measurements are shown in form of length overall, L_{OA}. These dimensions are set according to ITTC standard. [10].

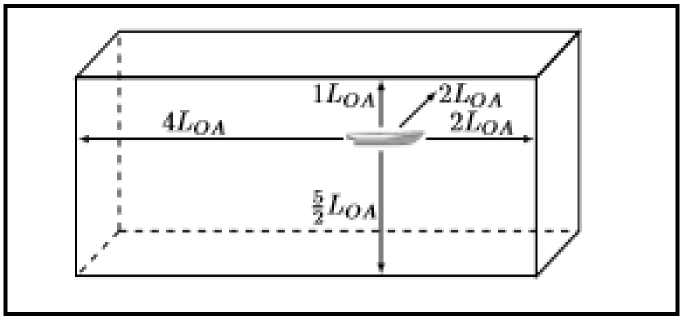


Figure 2: ITTC standard Computational domain

The CAD model of hull was designed. Details of design & process is given later in this section. After importing the hull geometry to the software, the hull geometry was subtracted from the rest of the domain. The domain was established as fluid flow. The undisturbed free surface level was later defined in fluent as per draught of the body.

B. Hull Dimensions

1) Athena Hull

Athena hull is semi-planing hull and following are the hull dimensions of Athena which are given Jenkins report.

Table 1: Dimensions of Scaled Athena Hull

Description	**Value**
Lpp	5.86m
B	0.836m
D	0.183m
S	4.22 m^2

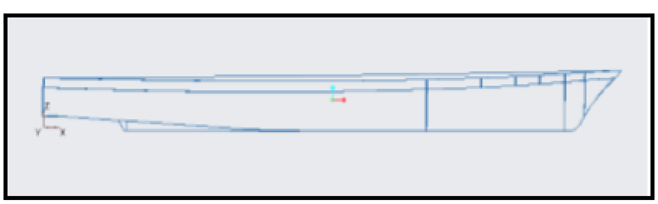


Figure 4: Isometric View of Athena Hull

2) Test Hull

Test Hull is also a Semi-planing Hull. Its Length overall, LOA, 39m. The simulation for test hull was conducted on Full scale, the dimensions of the test hull and simulation are given below,

Table 2: Dimensions of Test Hull

Description	**Value**
Lpp	39 m
B	8 m
D	1.90 m
S	4269.846 m^2

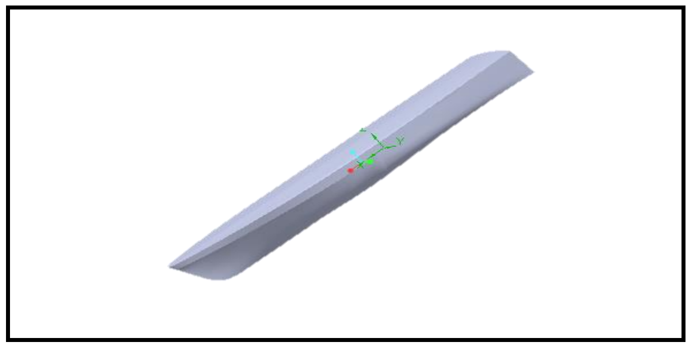


Figure 4: Isometric View of Test Hull

C. Mesh generation

The mesh density was focused on the areas in which flow phenomena is important to capture. A finer mesh was used on the hull surface and an inflation layer was also generated. This was done to determine the boundary layer around body and capture the shear stresses acting on hull correctly. The inflation layer was generated in accordance to the total thickness of boundary layer thickness. And first cell height for boundary layer was estimated to obtain a proper value of wall Y+. The high mesh density was used to capture the free surface accurately. The mesh in this region is very important because the hull induced waves occur in this region. A uniform cell height was used to avoid spreading of free surface in front of the hull. Other regions were covered with coarser mesh.

The mesh was generated using the Ansys meshing. The domain mesh comprises of the two regions; one is inner region and it is unstructured mesh based on tetrahedral cells and outer region is based on structured mesh. Interface between both regions was made non-conformal in order to keep the cell count low. Moreover, the grid dependence study to utilize a mesh with computationally low expense and relatively accurate result is done on the basis of formula updated for unstructured mesh [11] and is dependent upon the cube root of cell counts.

$$r_r = \sqrt[3]{\frac{N_1}{N_i}}$$

where

N_1 and N_i

total no. of cells in grid 1 and.grid *i*.

Structure of the mesh is shown below.

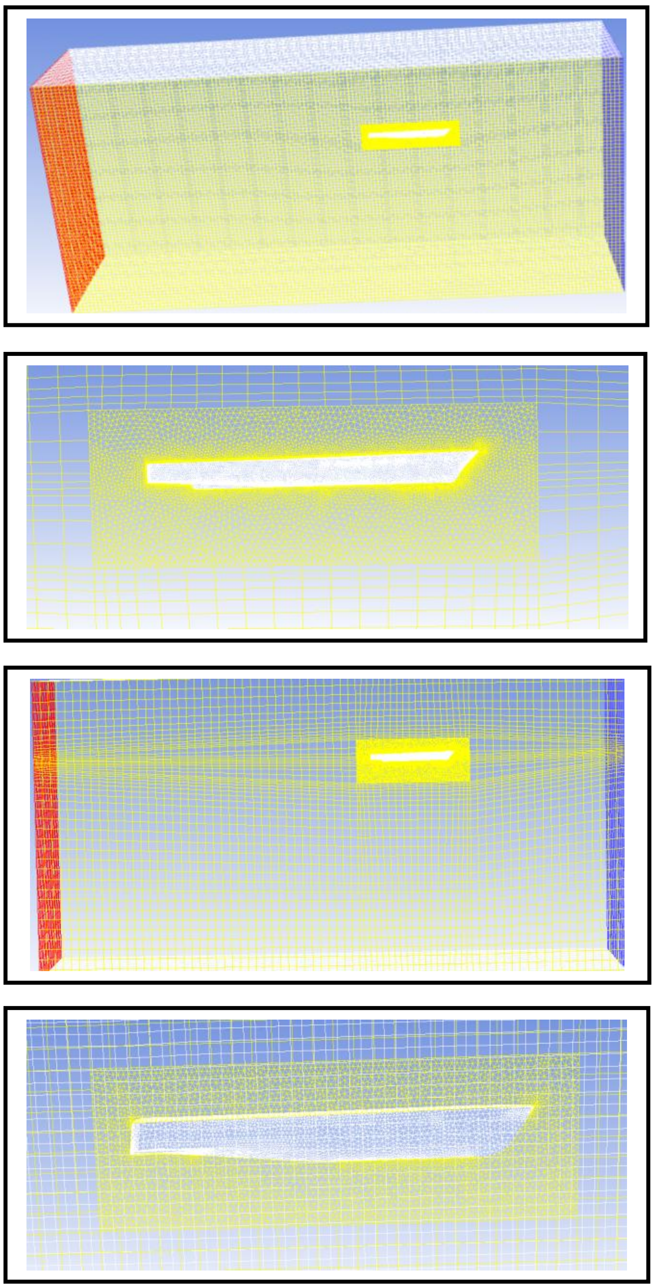


Figure 5: Mesh Structure

D. Model Definitions and properties

1) Mathematical models

Appropriate models were selected for simulations based on theoretical knowledge stated in the chapter. The interaction between hull and two-phase flow is well described by these models.

a) Two-phase flow models

Using RANS equations SST K-ω was chosen to model the turbulence because it captures the turbulence near hull and wake pattern of hull better than other models. And it is also good for high pressure gradients. Wall functions are used to avoid the resolution of whole domain. Fluent blended wall functions were used to meet the requirement. Low and high values of Y+ was used to get a suitable range of wall y^+ values. The thickness of first layer was set to get a suitable wall y+ value.

To model and resolve the free surface the VOF method was used. To validate the results the fluid properties was used same as in experiments also both water and air were understood as incompressible fluids.

b) Hull motion

The 6 DOF solver was enabled to resolve the equations of motion and rotation. To specify moment of inertia and mass of the hull model CAD software was used. Since only dynamic lift of the hull was simulated. The 6DOF solver was made limited to 2DOF. By allowing translational motion in Z-axis heave can be captured while the pitch can be captured by allowing the rotational motion about Y-axis. This whole setting was made by incorporating a UDF. The UDF is looped with the simulation to run with each time step.

2) Numerical methods

After selecting the mathematical models, numerical methods are finalized. Second order upwind scheme was selected for all the convection terms not including the Volume fraction (VF) equations to solving the spatial discretization schemes. In Fluent Compressive scheme is used for such simulations. The central differencing scheme is used to discretize the diffusion terms. Since temporal precision was not important, in temporal discretization the implicit scheme is chosen. SIMPLE algorithm is used for pressure-velocity coupling.

Diffusion based smoothing along with a parameter of 1.5 and remeshing was used for modeling of mesh in Fluent.

E. Boundary Conditions

Required boundaries conditions of the computational domain were set into Fluent. Computational domain boundaries are shown in the Fig given below.

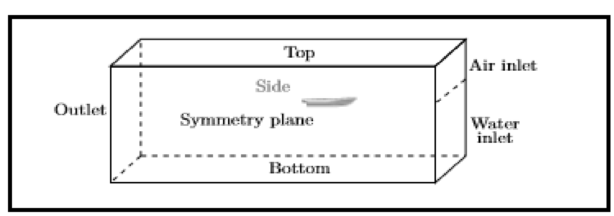


Figure 6: Domain

In Open Channel multiphase flow, the inlet was set as Pressure Inlet. While outlet is specified as Pressure outlet. By specifying the values for turbulent intensity and viscosity ratio all the variables are set. The free surface is located with the help of open channel flow.
Operating pressure was set as atmospheric pressure while Reference Pressure Location was given well above in the less dense region i.e. Air.
The Volume fraction was maintained by patching the regions i.e. Water and Air to the Computational domain.

V. Results & Discussion

The results obtained from CFD simulations will be discussed in this section. The results are distributed into two parts, Results of simulation for R/V Athena Hull, Results for Test Hull.

A. Athena Hull

In this section the results obtained from empirical techniques and the simulations are presented. The simulations were performed at different Froude numbers. First the Grid dependence study was carried out on a Froude to finalize the grid for further simulations. The calculated results are compared with experimental results and further discussion is done.

1) Grid dependence study

This study is done for Fn = 0.65. In grid dependence study the mesh was systematically refined in such a manner that 5 different meshes were obtained. The coarsest mesh had 2.58 million cells while the finest mesh had 8.04 million cells. The observation was carried out considering the total resistance coefficient convergence. It is represented in the graph given below. It is noticed that the 3rd mesh having 4.46 million cells is better tradeoff considering the accuracy and need of computational resources.

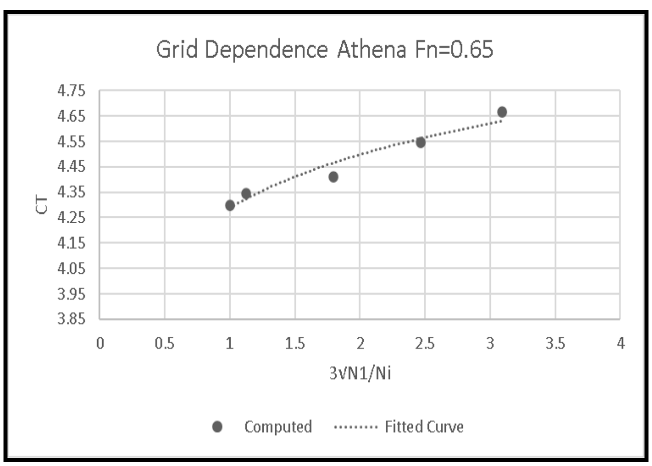


Figure 7: Total resistance coefficient convergence with Grid refinement

2) Volume Fraction of water at the hull

In the figure given below, contour plots of the volume fraction of water on the hull body is shown. The mesh was made very fine near the hull body. Therefore, the waterline is very clear. Here in this section the volume fraction of water at different Froude numbers will be shown.

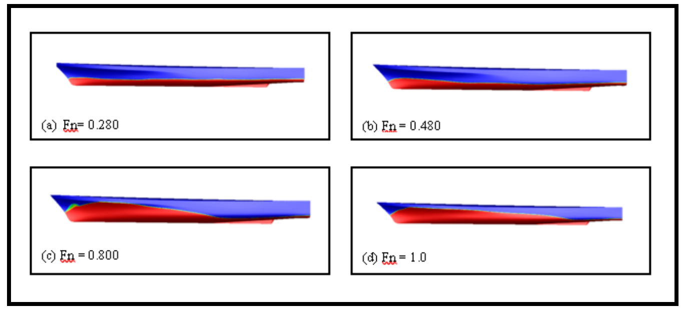


Figure 8: Volume Fraction at different Froude Numbers for Athena Hull

3) Dimensionless wall distance

This section shows the contour plots of dimensionless wall distance. The fist layer thickness of the prism layer was not set in a way that the region of air could be taken into account. The values of y^+ are not near to objective value, where the air interacts with the hull. But as the air resistance has less part in total resistance therefore this deviation is little important. The Y^+ values for water region was into the desired range described before.

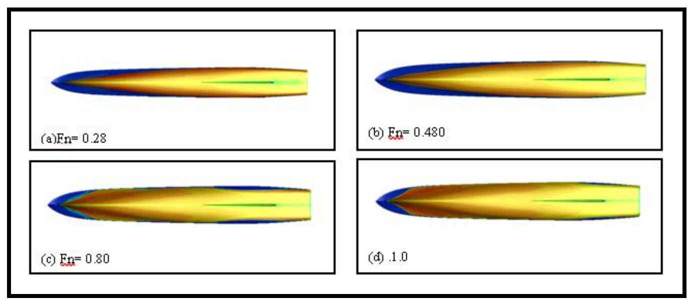


Figure 9: Wall Y plus contours at different Froude Numbers for Athena Hull

4) Pressure Coefficient

Contours plots of pressure coefficients are shown in the figures given below. A place of high pressure is observed where the water hits under the hull.

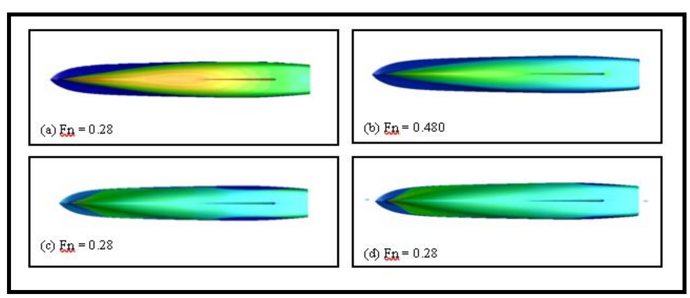


Figure 10: Pressure Coefficients at different Froude Numbers for Athena Hull

5) Free surface wave pattern

In the figure given below an iso-surface is generated to present the free surface. The free surface is classified as the air-water interface. The iso-surface is created where the volume fraction is 0.5. The wave pattern generated by the Fluent at Froude number 0.65 is thought to be good show.

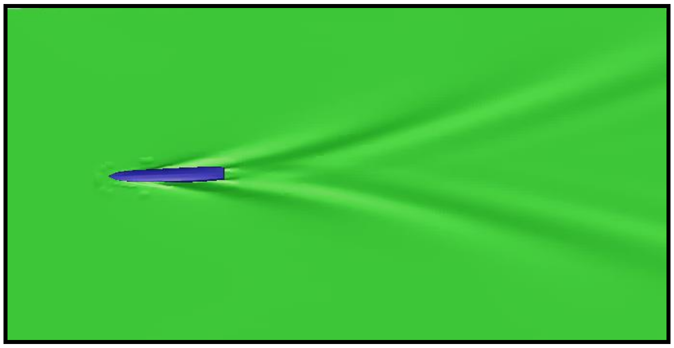


Figure 11: Wake pattern generated by Fluent for Athena Hull

6) Calculated Properties

The calculated results are compared with experimental results. It is seen from the simulations that the resistance coefficient is overpredicted. But the deviation remains within limit of 10%. The results are shown in the figure given below.

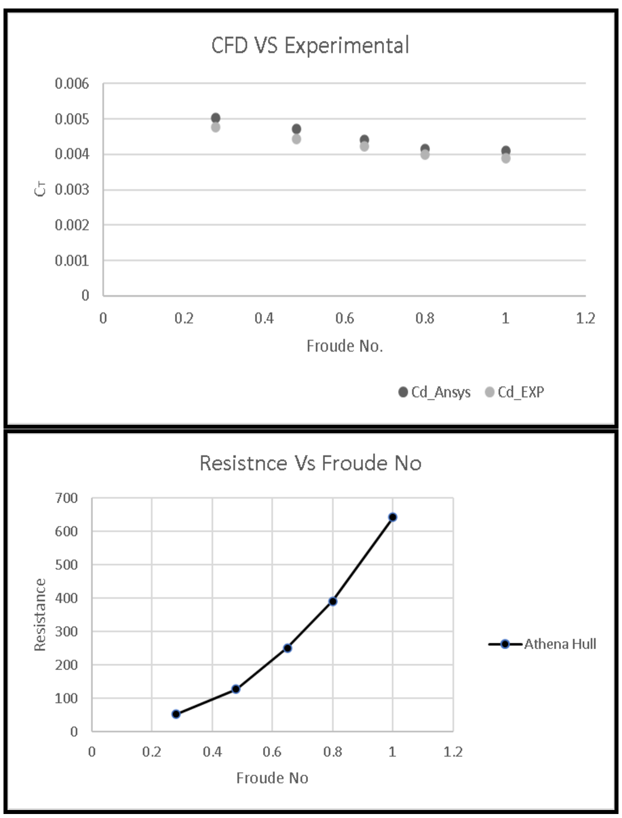


Figure 12: Comparison for validation of CFD Results & Resistance Curve of R/V Athena

The %error is shown in the figure given below. The error lies within the limit of 10%. It was analyzed the overprediction is generated from the sides of the hull. The empirical results for DTMB-5365 R/V Athena hull were driven using Savitsky method & Blount-Fox Method. The results from both methods comes into good agreement to CFD results as shown in the figure given below.

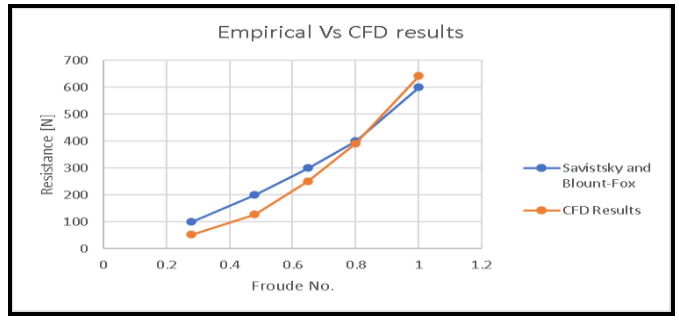


Figure 13: Comparison between empirical and CFD results

B. Test hull

In this section the results obtained from empirical techniques and the simulations are presented. The simulations were performed at different Froude numbers. The calculated results are compared with empirical results and further discussion is carried out.

1) Grid dependence study

After validating the technique with experimental results on Athena Hull. The test hull simulation was modelled on full scale. The mesh was made on the same technique but with proportionally larger cell size such that no. of cells in mesh generated are same. The Grid dependence study was not carried out for Test hull. Because it is deduced from the mesh which was generated incorporating the same method used for Athena hull but with proportionally larger cell sizes. The mesh generated through this technique had the same cell count that was generated for Athena hull. It produced satisfactory results which will be discussed in this section later. On basis of those results a hypothesis is made that a mesh valid for model geometry can be implemented on full scaled geometry with proportionally larger cell sizes.

2) Volume Fraction of water at the hull

In the figure given below, contour plots of the volume fraction of water on the hull body is shown. The mesh was made very fine near the hull body. Therefore, the waterline is very clear. Here in this section the volume fraction of water at different Froude numbers will be shown and discussed.

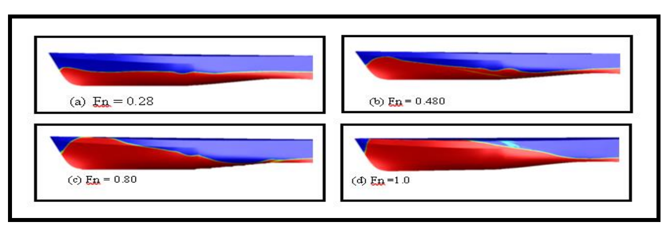


Figure 14: Volume Fraction at different Froude Numbers for Test Hull

In the figures shown above. It can be seen that water is splashing above the height of hull at high Froude numbers. This can be avoided by optimizing the geometry.

3) Dimensionless wall distance

This section shows the contour plots of dimensionless wall distance. The fist layer thickness of the prism layer was not set in a way that the region of air could be taken into account. The values of y+ are not near to objective value of y+, where the air interacts with the hull. But as the air resistance has less part in total resistance therefore this deviation is little important. The Y+ values for water region was into the desired range as described before.

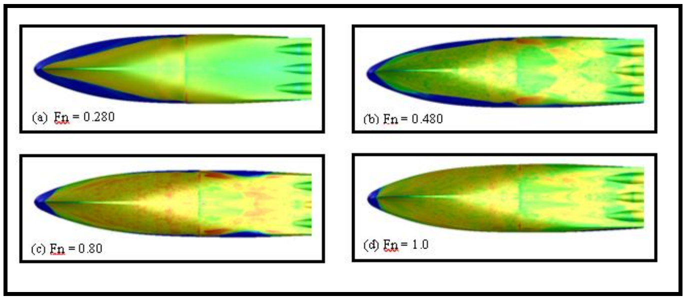


Figure 15: Wall Y+ contours at different Froude Numbers for Test hull

4) Pressure Coefficient

Contours plots of pressure coefficients are shown in the figures given below. A place of high pressure is observed where the water hits under the hull. During higher velocities the pressure coefficient is higher at sides of hull. The reason behind this is unknown but it means a keen attention is needed in this region.

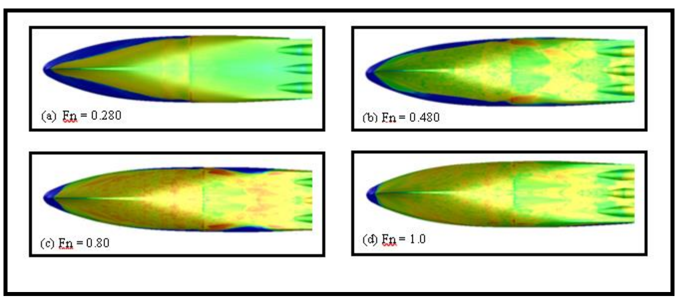


Figure 16: Pressure Coefficients at different Froude Numbers for Test Hull

5) Free surface wave pattern

In the figure given below an iso-surface is generated to present the free surface. The free surface is classified as the air-water interface. The iso-surface is created where the volume fraction is 0.5. The wave pattern generated by the Fluent at Froude number 0.65 is thought to be good show.

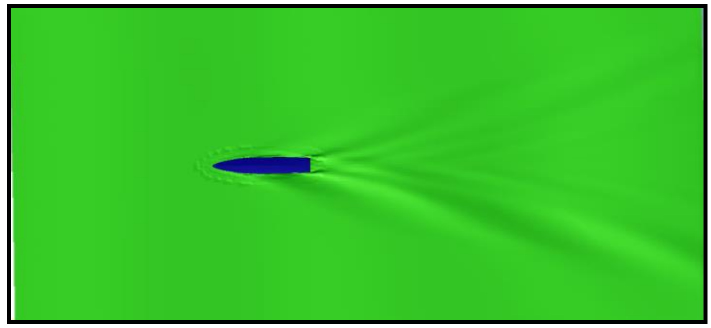


Figure 17: Wake pattern generated by Fluent for test hull

6) Calculated properties

In this section the results derived from the CFD simulation and empirical formula are shown and discussed. The drag force calculated from the CFD simulations is shown in the figure given below.

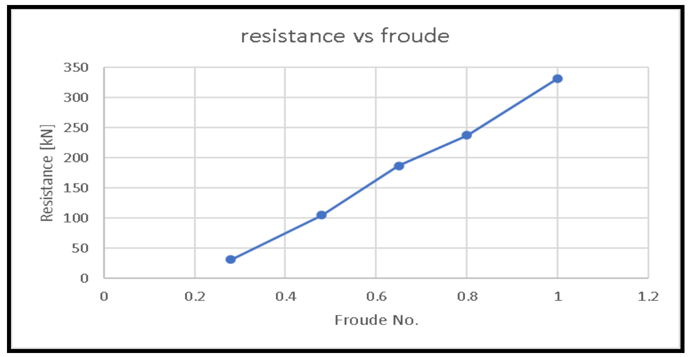


Figure 18: Resistance calculated from CFD at different Froude Numbers.

The empirical results were driven using Savitsky method & Blount-Fox Method. The Blount-Fox method overpredicted the Drag force the reason for this is the correction factor added into the formula specially for the planing hull. Savitsky method lies within good agreement to CFD results as shown in the figure given below.

For Froude number 0.28, Holtrop-Mennen method was used to predict the force because it lies in the displacement region and Savitsky method or Blount-Fox method is not made for this region.

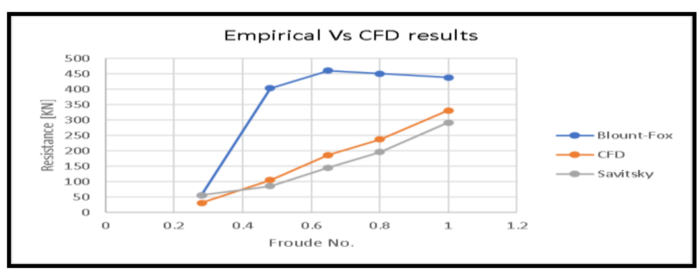


Figure 19: Comparison between empirical and CFD results

C. Dynamic mesh results

There could be a significant difference in drag force in case of planing hull being free to sink and trim. But it has been noted that for semi-planing hulls there is no big difference in the drag force in simulations free to sink and trim. The only important thing in our case is to notice the trim angle. When simulating the hull for free sinkage and trim in Fluent, it was hard to obtain an equilibrium position. It is observed that it was because of unphysical motion as the time-step chosen was not suitably small. After the minimizing the time-step the simulation however stabilized but after a large of number of iterations the simulation generated the negative cell volume. It was because of unphysical motions due to large lift forces at high speeds. Hull starts oscillating with increased amplitude and rapid changes occur in free surface. A keen mesh study was carried out to counter the error. It worked but stable solution for trim angle was not achieved. Another issue was the unavailability of experimental results for trim angles. So, there was no way to validate the results. The results driven from the simulation is shown in the graph given below.

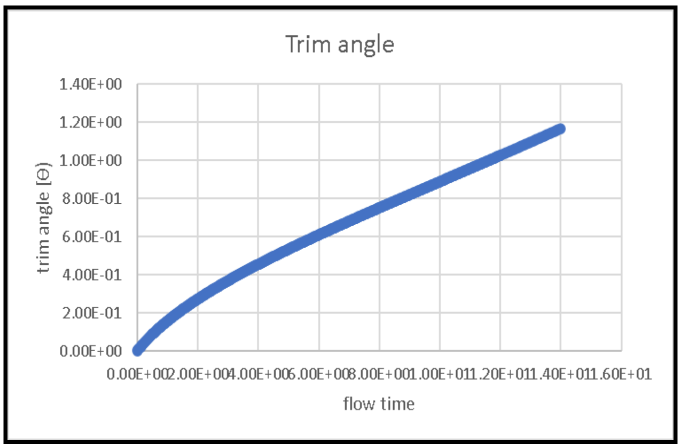
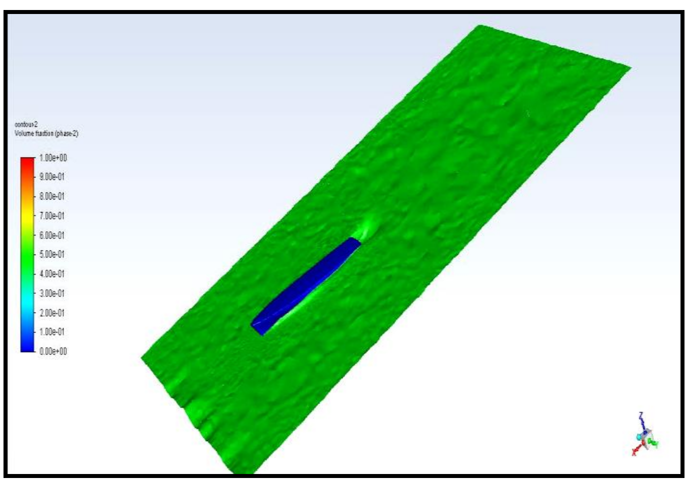
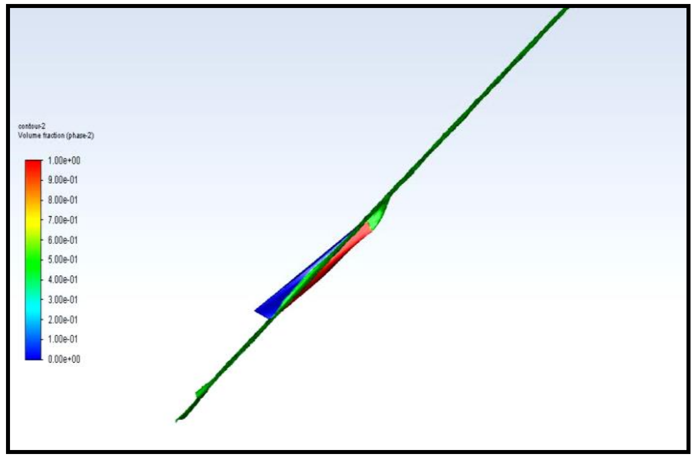


Figure 20: Unstable Trim Graph



(a)



(b)

Figure 21: (a) Dynamic lift View 1 (b) Dynamic lift View 2

The graph shows the instability of the trim angle. It is continuously increasing. It has also been seen in different studies that the reason behind being unable to obtain stable solution of fast boat in dynamic meshing is not known perfectly. This problem is also highlighted in the research study of David Frisk, Linda Tegehall,2015.

D. Geometry improvement

In this section some positions for the placement of lifting and spray rails will be suggested. A sculpted lifting rail (LR) could be designated at upper part of hull. While there should be two spray rails attached to the hull body. A short one SR1 should be designated near the stem below the long one, designated as SR2. In this way by means of SR1 the wetted area behind the stems will be minimal. And this is the main reason to suggest the SR1 here because it will leave SR2 unloaded. Because front part of spray which rises at rear part of stem will be thrown away before reaching SR2. In this way the risk of overflowing this rail could be minimized. The spray rails should have a triangular cross-section to enable separation of spray a during large break-off angles. The wetted area of round bilge hulls increases with high speeds. The spray rails play remarkable role in minimizing the wetted area during high speeds. The suggestions made so far are shown in figures given below.

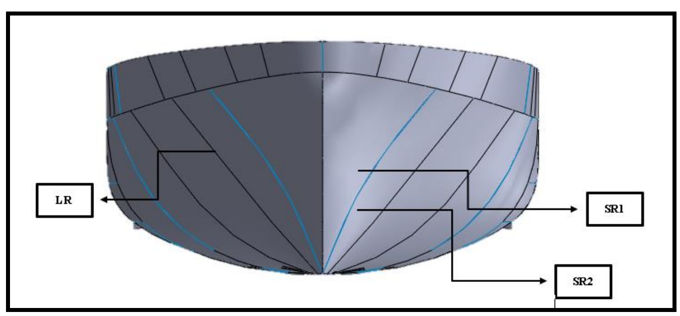


Figure 21: Suggestions for Geometry Improvement

These suggestions are made on the basis of study done by muller,1991. The effects of spray rails on semi-displacement hulls by Muller can be studied for implementing these suggestions. Moreover, a CFD analysis can be carried out to evaluate the performance efficiency.

VI. CONCLUSION

Empirical formulae can give a quick and crude estimate of hull. It can be useful in very early design stages like initial sizing etc. The resistance prediction methods applied in this thesis are Savitsky and Blount-Fox method. After getting the results from calculations, the comparison was carried out. It can be seen that the Savitsky method predicts low resistance in (Pre-planing). However, the Savitsky give better and more acceptable results in high speeds i.e. Semi-planing and planing modes). The results of Blount-Fox were over-predicted which may be because of the correction factor used for the resistance prediction of planing hulls.

It is very difficult to obtain data such as streamlines around hull and force distributions on the hull body from experiments, but CFD simulations can provide such data. Therefore, CFD can be an effective tool to be used by shipyards for testing and improving at early stage in design process. It is not the only matter of interest that CFD can replace towing tank or not based on accuracy of the results. It is also taken into the account that either simulations be faster or cost effective as compared to experimental analysis.

The results from CFD simulations in Ansys Fluent are in good agreement with experimental results. The resistance is overpredicted, but results deviate within the limit of 10%. The difference in pressure resistance is thought to be originated from the sides of the hull where a higher-pressure coefficient is predicted by Fluent. The CFD results were also compared to Empirical results which come into a good agreement.

VII. FUTURE WORK

The future work should focus on carry forwarding the simulation with different parts on the hull that are suggested after analyzing the flow pattern around the hull body. The mesh distribution near the hull should be better so that the important phenomena like wave-breaking and spray should be captured. A comparative analysis could be taken into account between the resistance prediction of bare hull and the hull with suggested body parts. Moreover, it would also be of interest to examine the maneuverability and stability of hull.

s

REFERENCES

[1] Holtrop, J., Mennen, G. G. J. A "Statistical Power Prediction Method", International Shipbuilding Progress, 25(290):253–256, 1978.

[2] Savitsky, D., Brown, P. W. "Procedures for Hydrodynamic Evaluation of Planing Hulls in Smooth and Rough Water", Marine Technology, 13(4):381–400, 1976.

[3] Brizzolara, S., Serra, F. "Accuracy of CFD Codes in the Prediction of Planing Surfaces Hydrodynamic Characteristics", 2nd International Conference on Marine Research and Transportation. Ischia, 2007.

[4] Azcueta, R. "Computation of Turbulent Free-Surface Flows Around Ships and Floating Bodies", PhD thesis. Hamburg: Technical University of Hamburg; 2001.

[5] Larsson, L., Stern, F., Visonneau, "Numerical Ship Hydrodynamics - An Assessment of the Gothenburg 2010 Workshop", Dordrecht: Springer, 2013.

[6] Brizzolara, S., Villa, D. "CFD Simulations of Planing Hulls. 7th International Conference on High-Performance Marine Vehicles", Melbourne, 2010.

[7] Visone, M., Bertetti, P., Gandolfi, R., Falletta, C., Ausonio, P. L., Paterna, D., et al. "VOF-Dynamic Mesh Simulations of Unsteady Ship Hydrodynamics", Marine CFD 2005 - 4th International Conference on Marine Hydrodynamics. London: The Royal Institution of Naval Architects, 2005.

[8] Caponnetto, M., Bučan, B., Pedišić-Buča, M., Perić, M., Pettinelli, C. "Simulation of Flow and Motion of High-Speed Vessels. Proceedings of the 12th International Conference on Fast Sea Transportation", Amsterdam, 2013.

[9] Fu, T.C.,Brucker,K.A., Mousaviraad,S.M., Ikeda, C.M.,Lee, E.J.,O'Shea, T. T., et al. "An Assessment of Computational Fluid Dynamics Predictions of the Hydrodynamics of High-Speed Planing Craft in Calm Water and Waves", 30th Symposium on Naval Hydrodynamics. Hobart, 2014.

[10] 25th ITTC Resistance Committee, "Uncertainty Analysis in CFD: Verification and Validation Methodology and Procedures. Technical report 7.5-03-01-01", Revision 02. International Towing Tank Conference (ITTC), 2008.

[11] Roache, P. J. Perspective, "A Method for Uniform Reporting of Grid Refinement Studies" Journal of Fluids Engineering, 116(3):405–413, 1994.

[12] ANSYS Fluent Theory Guide. Release 16.0. ANSYS. 2015.

[13] ANSYS Fluent User's Guide. Release 16.0. ANSYS. 2015.