October 2012

Computational Estimation And RANS Simulation of Free Surface Flow Around A Ship Hull

Katuri Samarpana

M.V.G.R. Engineering College/ Mechanical, Vizianagaram, India, katurisweety@gmail.com

Follow this and additional works at: <https://www.interscience.in/ijmie>

Part of the [Manufacturing Commons](#), [Operations Research](#), [Systems Engineering and Industrial Engineering Commons](#), and the [Risk Analysis Commons](#)

Recommended Citation

Samarpana, Katuri (2012) "Computational Estimation And RANS Simulation of Free Surface Flow Around A Ship Hull," *International Journal of Mechanical and Industrial Engineering*: Vol. 2 : Iss. 2 , Article 9.

DOI: 10.47893/IJMIE.2012.1079

Available at: <https://www.interscience.in/ijmie/vol2/iss2/9>

This Article is brought to you for free and open access by the Interscience Journals at Interscience Research Network. It has been accepted for inclusion in International Journal of Mechanical and Industrial Engineering by an authorized editor of Interscience Research Network. For more information, please contact sritampatnaik@gmail.com.

Computational Estimation And RANS Simulation of Free Surface Flow Around A Ship Hull

Katuri Samarpana

M.V.G.R. Engineering College/ Mechanical, Vizianagaram, India
E-mail : katurisweety@gmail.com

Abstract - Ship hydrodynamics present many unique challenges due to complex geometry, environment, and operating conditions, which results in many complex physics and modelling issues. This is commonly studied through experiments in a towing tank and experiments in a sea keeping and manoeuvring basin. Recently hydrodynamicists have begun to venture into computational prediction of hydrodynamic behaviour of surface ships. Free surface phenomenon around a ship hull plays an important role in its resistance. Wave making resistance comes from the very presence of free surface. Therefore its accurate prediction is very essential for ship design. The flow problem to be simulated is rich in complexity and poses many modelling challenges because of the existence of breaking waves around the ship hull involving two-phase flow, and because of the resolution of thin turbulent boundary layer. The paper aims to computationally estimate the effect of free surface for a moving ship. Commercial software is used for grid generation and flow solution.

1. Solution of a Rudder of a ship in submerged condition.

Few different shapes of the rudders are examined.

2. Solution of flow- around a complete ship with free surface.

In the present work, flow through the ship hull is computed using a finite volume commercial code, ANSYS 12.1. The ship geometry is modelled using solid modelling software, CATIA V5R9. A three-dimensional structured hexahedral grid is generated using grid generating code, ICEM-CFD V10.0. Turbulence is modelled with Reynolds Stress model. The resistance of the ship is predicted, and compared against the experimental values. The rudder of the ship is also analyzed. Two different shapes, one wedge shaped and a standard NACA0012 foil, for which experimental results are available in literature, are analyzed. The lift coefficients and flow separation are predicted for different angles of attack using various turbulence models. Computational results are in good agreement with the experimental ones.

I. INTRODUCTION

Advances in computing technology, software and hardware have revolutionized the design process of engineering vehicles such as aircrafts, automobiles and ships. Many commercial software packages are being used in the design as well as analysis processes which not only save lead time and costs of new designs, but also are used to study systems where controlled experiments are difficult or impossible to perform. In the area of fluid dynamics, there are many commercial computational fluid dynamic (CFD) packages available for modelling flow in or around objects. Computational fluid dynamics (CFD) has been constantly developed over the past few decades and now both commercial and research codes can provide more and more robust and accurate results. The waves have a big impact on the forces acting on the hull. As the shipbuilding industry moves to design hull forms that run faster and generate

less noise, computer simulation is playing an increasingly important role.

The current study aims to capture the free surface flow around a ship hull and analysis of its rudder. Solution of flow around the rudder involve only one phase of liquid i.e., water and was solved first. Results were verified with the experimental data. A NACA foil was analyzed as an alternative shape. Both the shapes of the rudder, wedge and NACA0012 foil are analyzed at different angles of attack from 0° to 20° using various turbulent models and the results are in good agreement with the experimental ones.

II. OBJECTIVES

The objective of the present work is to

1. Validate the CFD results of NACA0012 against experimental data using different turbulence models and schemes.

2. Establish a numerical technique to simulate flow around the ship hull and estimate the resistance and compare the results with the experimental ones.

This aims to predict the following:

- Estimate the lift of NACA0012 and wedge section.
- Estimation of Pressure and viscous resistance of the ship.
- Capture the free surface around the ship hull
- Flow visualization

III. COMPUTATIONAL METHODS

Computation fluid dynamics mainly depends upon three fundamental governing equations of fluid dynamics

- Mass conservation (Continuity equation)
- Momentum Conservation (Navier-Stokes Equation)
- Energy Conservation (Energy equation)

The various complex and interesting flows result form the solution of these equations subjected to different physical boundary conditions.

The numerical solution of these equations subjected to the boundary conditions can be obtained by the use of the following methods:

1. Finite Difference Method (FDM)
2. Finite Element Method (FEM)
3. Finite Volume Method (FVM)

There are a variety of other methods but the three above are mostly used in the fields of CFD research. In this work, we have used the commercial software, “FLUENT” which uses the FVM. In the following paragraphs, the FVM is very briefly described.

Finite Volume method often called the Control Volume Method is formulated from the inner product of the governing differential equations with a unit function. This process results in the spatial integration of the governing equations. The integrated terms are approximated by either finite differences or finite elements discretely summed over the entire domain.

One of the most important features of the FVM is its flexibility for unstructured grids. Designation of the components of a vector normal to boundary surfaces in FVM accommodates the unstructured grid Configuration with each boundary surface integral constructed between nodal points.

The principle equation in the CFD models for general viscous flow may be represented by the time averaged Navier-stokes equation.

IV. CFD FLOW

The fundamental method for creating a CFD model can be represented as a “pipeline” of six steps as illustrated in Figure below.

The process begins with the creation of an assembly of one or more parts (geometric shapes) that reflect the design under consideration. Because CFD deals with the flow through or around solid objects, the CFD model is built upon the inverse of the assembly and is referred to as the “Flow Volume Extract”. This Flow Volume Extract is meshed to specification, a simulation is performed, and finally, the results of the simulation are analyzed. Because the CFD model deals with the inverse of the solid assembly, it is important that the original geometry is created using surfaces that are connected, with faces that about one another perfectly. Geometric models containing “cracks”, “holes”, or “gaps” will create problems farther down the CFD pipeline, at the meshing stage. Therefore, it is important to insert a geometry check stage into the process. This stage performs an analysis of the initial geometry to ensure that it does not contain features that will cause a failure later on. Geometric features that do not pass these checks are flagged and labelled for repair within a geometric editing environment, such as those found on popular CAD and PLM systems and in software supplied by Fluent. Fluent provides with a full range of tools to perform successful CFD analyses and meet your individual needs and goals.

V. THEORETICAL CALCULATIONS OF SHIP RESISTANCE:

Ship resistance can be identified as reaction to the ship hull from water. From the experimental and numerical methodology, it can be said there are three components of the ship resistance:

1. Wave Resistance (R_w)
2. Viscous Resistance (R_f)
3. Viscous pressure Resistance (R_{vp})

1. WAVE RESISTANCE:

A ship makes waves when it sails. In this wave energy is stored. This energy is the wave making resistance. A wave is produced by the curvature of a ship. Around a ship changing the direction of the water results not only in a pressure difference, but also in a change of water level. The more and the sharper the water is changed in direction, the bigger the waves. Positive pressure results in a higher level, negative pressure results in a lower level. Typical high pressure is built at the bow, where the water is deflected from the hull.

USING DIMENSIONAL SIMILARITY:

$$\frac{V_m}{\sqrt{L_m}} = \frac{V_p}{\sqrt{L_p}}$$

$$V_m = \sqrt{\frac{L_m}{L_p}} * V_p$$

$$V_m = \sqrt{\frac{150}{150000}} * 18$$

$$V_m = 0.569 \text{ m/sec}$$

$$R^* = \frac{R_w}{\frac{8}{\pi} \rho g B^2 \frac{T^2}{L}}$$

$$R^* = \frac{R_w}{\frac{8}{\pi} * 1027 * 9.81 * 0.0162^2 * \frac{0.004^2}{0.15}}$$

$$R^* = \frac{R_w}{2.855 * 10^{-3}}$$

As wave resistance coefficient

$$C_w = \frac{R^*}{F_n^2}$$

Where F_n is Froude number

$$F_n = \frac{v}{\sqrt{Lg}}$$

$$F_n = \frac{0.569}{\sqrt{0.15 * 9.81}}$$

$$F_n = 0.47$$

Substituting F_n and R^* in wave resistance coefficient equation

Considering $C_w = 0.796$

$$C_w = \frac{R^*}{F_n^2}$$

$$0.796 = \frac{R_w}{9.089 * 10^{-4} * 0.47^2}$$

$$R_w = 1.589 * 10^{-4} \text{ N}$$

Similarly Viscous resistance (R_f) and Viscous pressure Resistance (R_{pv}) can be calculated.

S.NO	PARAMETERS	SCALED MODEL	UNSCALED MODEL
1	LOA	150mm	150m
2	BREADTH	16.2mm	16.2m
3	DRAFT	4.5mm	4.5m
4	VELOCITY	0.569m/sec	18m/sec
5	FROUDE NUMBER	0.47	0.47
6	WAVE RESISTANCE(R_w)	$1.589 * 10^{-4} \text{ N}$	159.54KN
7	FRICTION RESISTANCE(R_f)	$1.84 * 10^{-3} \text{ N}$	505KN
8	VISCOUS PRESSURE RESISTANCE(R_{pv})	$1.8547 * 10^{-4} \text{ N}$	51.32KN
9	TOTAL RESISTANCE(R_t)	$2.184 * 10^{-3} \text{ N}$	715.83KN

VI. GEOMETRIC DETAILS OF WEDGE SHAPED RUDDER

The geometry of the wedge shaped rudder are described in figures 5.1 and tables 5.1 respectively. A 2D model is created in GAMBIT.

Fig 5.1 Geometry of Wedge Shaped Rudder

Length of the rudder	107 mm
Radius of the arc	7 mm
Velocity of rudder	25.72 m/s or 50 knots

Table 5.1 : Geometry Details of Rudder

Fig. 6.1 : Pressure coefficient around the body of Wedge shaped rudder at 0deg

VII. RESULTS AND CONCLUSIONS

From the present study, the following conclusions can be drawn:

1. CFD results of rudder were in good agreement with the computational values. Wedge shaped rudder produced more lift forces. Flow separation for wedge shaped rudder was also delayed. So this rudder was selected as final design.
2. By using of turbulence models and wall functions resulted in wide variation to the final prediction. Mesh fineness requires being compatible with the turbulence model/wall functions being used to obtain meaningful results.
3. CFD predictions for ship resistance find good agreement with the experimental results. The work validates the use of commercial CFD software to study free surface flows. However, more exercises have to be done in future to establish a better validation.

4. Literature survey reveals that at present, the subject of CFD estimation of free-surface flows has not reached to high levels of maturity. In view of this global status, present results are very encouraging.

The commercial CFD software FLUENT V6.3 was utilized for analysis. The VOF technique was used for capturing the free surface and Reynolds Stress

Model was adopted for turbulence closure. The mesh quality has proven to be an important issue in the computations. Future work should focus on employing a finer resolution grid than the one employed in present study. Different turbulence models can be tested in the simulations. Finer details of ship flow, as required by ship designers, like slamming, sea-keeping etc. can be studied in future.

REFERENCES

- [1] Ira H. Abbott & Albert E. Von Doenhoff "The subsonic aerodynamic characteristics of modern NACA wing sections"
- [2] Sighard F. Hoerner "Theoretical and practical information of skin friction drag and pressure drag".
- [3] Hirt, C. W and Nichols, B.D "Finite difference numerical simulations".
- [4] H. Yasukawa "Rankine Source Method to the Shallow water and channel Problems".
- [5] Majumber and B. N. Rajani "Computational Fluid Dynamics".
- [6] Shiotani.S and Kodama.Y "The finite-volume methods".
- [7] David C. Wilcox "Turbulence flow modelling".
- [8] Md. Akhtaruzzaman Sarker and David G.Rhodes "the volume of fluid free surface model experiments".
- [9] Marilyn J. Smith,[25] et al "Prediction of the lift, drag and pitching moments for rotor foils for the SC 1095 airfoil"