

Research Article

CFD Simulation and Analysis for Free Surface Computations around Fixed ‘DTMB 5415’ Model

Suyog S. Kadam^{A*}, Shivprakash B. Barve^A, Harshada A. Gurav^B, Vilas S Kanthale^A

^ADepartment of Mechanical Engineering, MAEER’s MIT College of Engineering, Pune, India

^BDepartment of Mechanical, Abhinav Education Society’s College of Engineering, Pune, India

Accepted 12 March 2014, Available online 01 April 2014, **Special Issue-3, (April 2014)**

Abstract

One important issue in the CFD application is the prediction of the power demand of a new ship. For this purpose the interaction between the hull, rudder and the propeller must be correctly accounted for. This project presents results of the computations performed in the ETSIN for different ships with the RANSE free surface commercial solver CFX. Some of the computational results are validated against experimental data in terms of various global and local quantities. The CFX code is based on a finite volume Discretization. The turbulence model used in the calculations was the SST (Shear Stress Transport) model, and the volume of fluid method is used to model the free-surface flow. The incompressible turbulent free surface flow around the complex hull form of the DTMB 5415 model at two different speeds has been numerically simulated using the RANSE code CFX. The Volume of Fluid method (VOF) has been used with CFX for capturing the free surface flow around the ship model at the two speeds. The simulation conditions are the ones for which experimental and numerical results exist. The standard $k-\epsilon$ turbulence model has been used in CFX code. The grid generator ICEM CFD has been used for building the hybrid grid for the RANSE code solver. The results compare well with the available experimental and numerical data.

Keywords: RANSE, Nodes & boundary conditions, Drag Force & drag Coefficient

1. Introduction

Fluid dynamics is the science of fluid motion. Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. Computational Fluid Dynamics is “a wind tunnel in the computer.” It is a method by which one uses certain algorithms or other numerical formulas to analyze the fluids' flow. It is one of the most important high tech tools for measuring the performance of model. Computational fluid dynamics (CFD) has progressed rapidly in the past 50 years. It has been used in many industrial fields and plays an irreplaceable role in engineering design and scientific research. CFD is the art of replacing the differential equation governing the Fluid Flow, with a set of algebraic equations (the process is called Discretization), which in turn can be solved with the aid of a digital computer to get an approximate solution. One important issue in the CFD application is the prediction of the power demand of a new ship. For this purpose the interaction between the hull, rudder and the propeller must be correctly accounted for. This paper

presents results of the computations performed in the ETSIN for US navy combatant DTMB 5415 ship with the RANSE free surface commercial solver CFX. Some of the computational results are validated against experimental data in terms of various global and local quantities. The CFX code is based on a finite volume Discretization. The turbulence model used in the calculations was the SST (Shear Stress Transport) model, and the volume of fluid method is used to model the free-surface flow. The numerical schemes use higher order cells to satisfy the momentum, pressure and turbulence quantities where each hexahedral cell is further subdivided into eight sub-volumes. A control volume is formed from the sub-volumes surrounding a grid node and a first order finite element basis function is used for each sub volume. The momentum and pressure are simultaneously satisfied using a coupled solution system. All the numerical equations are solved using algebraic multi-grid acceleration with implicit smoothing. Parallel computation on a 2 processors PC was adopted to reduce the required computational time.

Table 1 Terms & Nomenclature

Term	Meaning
Aft	stern or rear side

*Corresponding author: Suyog S. Kadam

ANSYS	The software used for CFD analysis.
Aspect Ratio	Ratio of longest edge length to shortest edge length. (Measure of quality of mesh)
BEM	Boundary Element Method.
CFD	Computation Fluid Dynamics. Calculation method or software which enables hydrodynamic optimization of hull form.
Draft	Depth of water to which a ship sinks according to its load. Vertical distance between ship's waterline and lowest point of its keel.
Drag	It is the resistance of the fluid to the moving body In longitudinal direction.
DTMB	David Taylor Model Basin.
FEM	Finite Element Method.
FDM	Finite Difference Method
Forward	front of ship
Froude number (Fn)	Froude number describes the vessel's relative speed, which depends on vessel length $F_n = \frac{v \left[\frac{m}{s} \right]}{\sqrt{g \left[\frac{m}{s^2} \right] \times L [m]}}$ Where, v = vessel speed in [m/s] (1 knot = 0.5 m/s) g = 9.81 m/s ² L = vessel's waterline length
FVM	Finite Volume Method
ETSIN CFD	computer program which calculate the steady inviscid flow around a ship hull, wave pattern and resistance
Hull	outer body of ship
RANSE	Reynolds Averaged Navier Stokes Equation
Resistance	Vessels resistance against moving eg. Wind resistance, friction resistance etc.
SST	Shear stress transport
VOF	Volume of fluid

2. Literature Review

[1] RANSE with free surface computations around fixed DTMB 5415 model

Martín Priego Wood¹, Leo M. González¹, Jorge Izquierdo¹, Adrián Sarasquete² and Luis Pérez Rojas¹(1 School of Naval Architecture, Polytechnic University of Madrid (Spain),2Baliño S.A.)

Different calculations have been done with unstructured and structured meshes obtaining qualitative and quantitative better results with the structured ones. This points out that although structured meshes are much more

complicated to create; they give much better results especially at the ship wake. Comparisons between experimental and computed values show a relatively good agreement between them using structured meshes. The structured mesh is based on a O-grid block distribution. The O-grid creation capability is simply the modification of a single block or blocks to a 5 sub block topology (7 sub-blocks in 3D) as shown in figure for a simple 2D case.
 ✓ The mesh quality can also be determined in ICEM. It should be positive. The quality is given in determinant form as follows
 ✓ Here, the number of cells are shown on Y axis while mesh quality is shown on X axis.
 ✓ A Determinant value of 1 would indicate a perfectly regular mesh element, 0 would indicate an element degenerate in one or more edges, and negative values would indicate inverted elements.^[1]

[2] Structure of CFD Code

Tao Xing and Fred Stern, IIHR— Hydro science & Engineering, C. Maxwell Stanley Hydraulics Laboratory, The University of Iowa.

2.1 Pre Processing

One limiting factor of the practical application of RANSE solvers lies in the model preparation (grid), which - is usually very time-consuming, thus using highly qualified manpower for low added value work, - and can hardly be automated for shape variations involved in an optimal design process. This explains the hard work performed by grid generation specialists to provide tools that reduce these issues. Such a software package is used in the present work which collects the advantages of being independent from solvers, including many user requested capabilities and having automation and scripting capabilities that make it directly incorporable into an optimal design procedure.

2.2 Solver

The offer in term of flow calculation tools is very large. However, when looking at ship specific issues, i.e. those involving a free surface, mainly two types of approaches are considered. The potential flow approaches based on boundary elements methods have now proven to be efficient for a number of basic problems like steady flow assessment (wave resistance problem) and response in waves (sea keeping problems). Many teams, from both commercial solver providers and maritime scientific community, are investigating the use of RANSE solvers for ship design problems. This is continuously leading to improvement in accuracy and extension of the range of application of these methods. However, only a small number of them can nowadays be practically involved in ship design, and they still require high level skills that prevent them to be currently used by most design offices. The Reynolds Averaged Navier-Stokes Equations discredited by finite differences, by taking into account the

complete nonlinear free surface conditions. Solver is dedicated to flow analysis around bodies piercing the free surface or close to it, and gives accurate prediction of all the components of ship resistance.

2.3 Post Processing

Another limiting factor to the use of RANSE methods in design procedure and especially in optimal design is related with the complexity of phenomena to be looked at, and with the difficulty to extract relevant figures of merit from the large amount of information provided by these solvers. Again, this aspect is worked on by many teams' specialists in this area. Several software packages are now commercially available for this purpose, among which the one used in the present work, ANSYS CFX. It gathers a large number of user required capabilities which enable a full exploitation of CFD results, and also has high automation and scripting capabilities which make it a good component of any optimal design approach. ^[11]

[3] Applied Computational Fluid Dynamics

Methodology by André Bakker.

Methodology includes-

- Modeling
- Meshing
- Defining boundary conditions
- Computational method
- Post processing

3. Procedure

3.1 Modeling

As we know, this analysis contains two types of fluids. Therefore "Multiphase modeling" is done. The modeling is done in ANSYS ICEM. This process involves step by step creation of geometry. First points are created and then they are joined by curves. Then surfaces are created and blocking is done. The detailed procedure is shown below

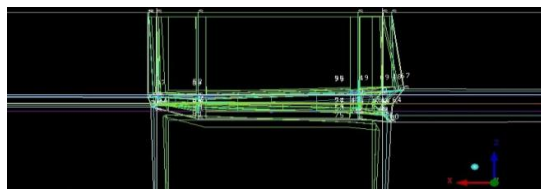


Fig.1 Blocking of Model

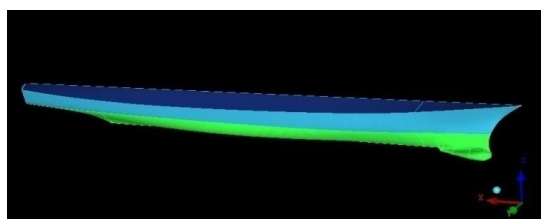


Fig.2 Actual Ship model in Ansys ICEM

3.2 Meshing

It is also called as Discretization. Domain is discretized into a finite set of control volumes or cells. The discretized domain is called the "grid" or the "mesh." General conservation (transport) equations for mass, momentum, energy, etc., are discretized into algebraic equations. All equations are solved to render flow field.

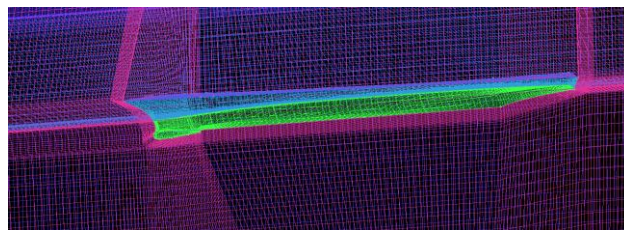


Fig. 3 Final meshed model of US navy Combatant DTMB 5415

The meshed model of ship shown above has following specifications.

Table 2 Mesh Details of Elemental Parts

Element parts	No. of elements/cells
AIR	1055208
GEOM	20892
INLET	3610
OPENING	44460
OUTLET	3610
SHIP BOTTOM	4832
SHIP TOP	3487
SHIP TOP1	1680
SYMMETRY	44868
WATER	1116430
WATER AIR INTERFACE	19212
WATER BOTTOM	17784

Total elements: 2020907; Total nodes: 1929333

3.3 Boundary Conditions

For the simplification of the problem, the reference states are selected while introducing the boundary conditions. Here these references are-

For ideal fluids

Reference temperature- 25°C

Reference pressure – 1 atm.

Reference index for radiation properties- 1

3.4 Post-Processing

In the post processing the results are reviewed in one of the two ways. Either graphically or alpha numerically. Graphically it shows contours, vector plots, charts, iso-surfaces, flow lines and animation. Numerically it gives Integral values, Drag, lift, and torque calculations, Averages, standard deviations, Minima, maxima, Comparison with experimental data.

For given test conditions, we are getting the results of following.

Graphical results

- Mesh quality checks
- Y plus plot
- Y plot
- Water & air velocity streamline
- Aspect ratio at interference
- Vector plots

Analytical results

- Drag force
- Drag coefficient

Table 3 Comparison between Experimental & Computational Values

Parameter	Experimental data	Computational data
Length (m)	5.72	5.72
Beam (m)	0.76	0.7242
Draft (m)	0.248	0.248
Wetted surface area (m ²)	4.486	4.861
Density (kg/m ³)	999	997

4. Results and Validation

4.1 Analytical results

Table 4 Analytical Results

Drag force	46.6928
Drag Coefficient	4.368

4.2 Graphical results

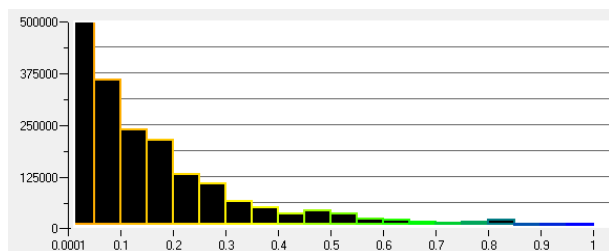


Fig. 4 Aspect Ratio Check

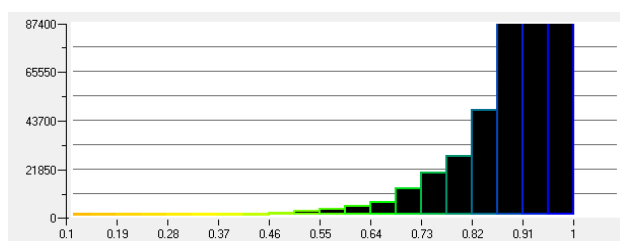


Fig. 5 Determinant check

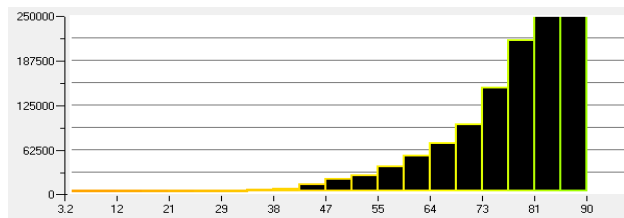


Fig. 6 Minimum angle check

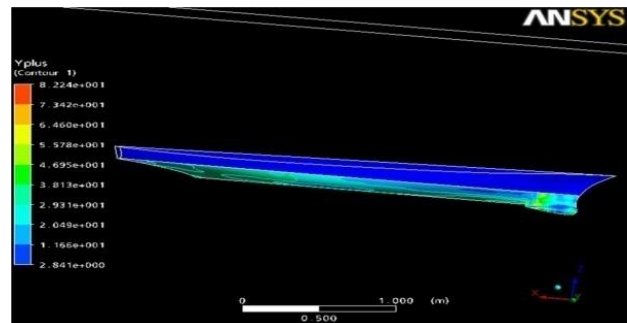


Fig.7 Y plus plot

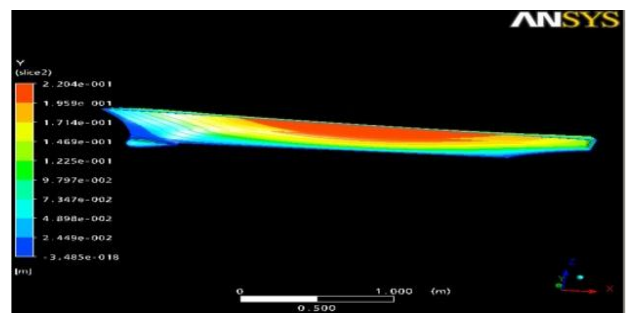


Fig. 8 Y plot

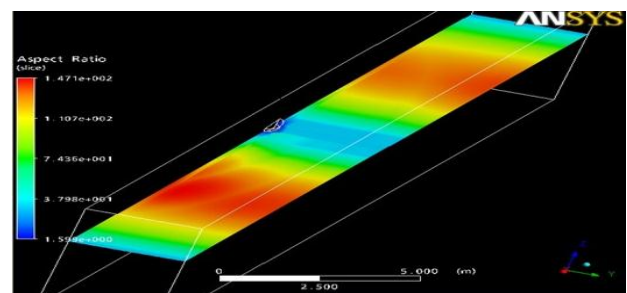


Fig. 9 Aspect ratio at interference

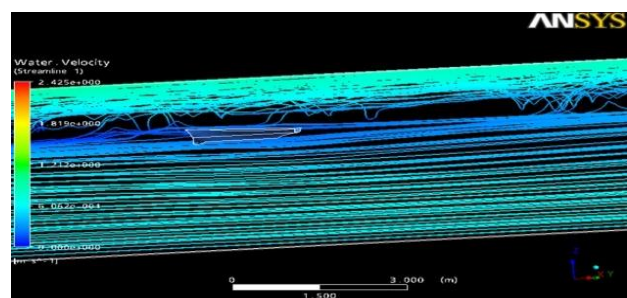


Fig.10 Water and air velocity streamlines

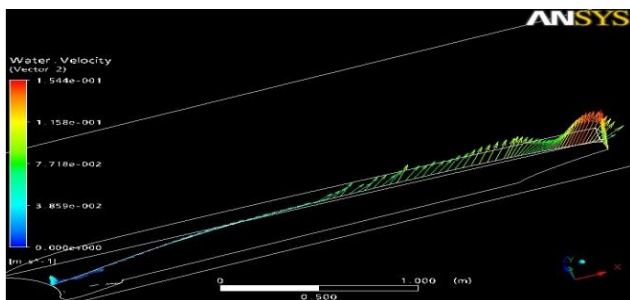


Fig.11 Water velocity vector plot at the interference

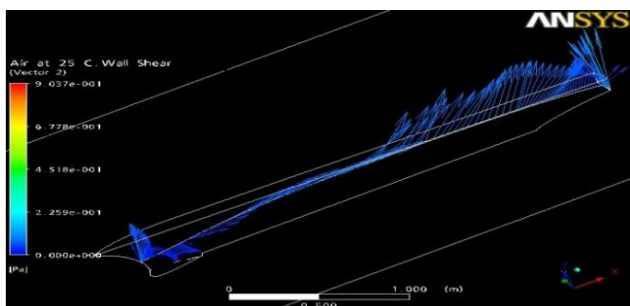


Fig. 12 Vector plot of air a 25°C wall shear

4.3 Validation

Table 5 Comparison between Experimental and Computational Results

	Experimental	Computational
Drag force	45.21 N	46.6928 N
Drag coefficient	4.23	4.368

$$\% \text{ Error} = (\text{Computational value} - \text{Experimental value}) / \text{Experimental Value}$$

The total resistance coefficient on the adapted mesh has the value **4.368**, which is **3.262%** higher than the experimental value.

The total resistance force on the adapted mesh has the value **46.6928**, which is **3.28%** higher than the experimental value.

Conclusions

A numerical study with the Ansys-CFX computational tool has been performed. A classical benchmark test case like the Combatant DTMB 5415 model has been investigated, which has been used for verification. Our computation was especially interested in the velocity field calculation at the ship hull. Other local calculations like contours, vector plots, mesh quality checks, drag force & coefficient values have been also obtained. Different calculations have been done with unstructured & structured meshes obtaining qualitative & quantitative better results with the last ones. This point's out that although structured meshes are much more complicated to

create; they give much better results especially at the ship hull. Comparisons between experimental & computed values shows relatively good agreement between them using structured meshes.

Future Scope

This analysis was only about "Drag" & "Mesh quality" but further using this technique we can also obtain following results,

- Fuel Economy
- Wake Coefficients
- Actual Velocity profiles & isolines
- Wave elevations
- Wave profile at required section
- Balancing, Stability & effect of waves on ship & stock on the ship
- Heating & cooling of engine
- Also results for other ship parts like Propeller

Further by changing the type of mesh, different results of different accuracy can be obtained.

References

Martín Priego Wood¹, Leo M. González¹, Jorge Izquierdo¹, Adrián Sarasquete² and Luis Pérez Rojas (2007), RANSE with free surface computations around fixed DTMB 5415 model and other Baliño's fishing vessels. *9th International Conference on Numerical Ship Hydrodynamics* Michigan, USA.

Tao Xing and Fred Stern, IIHR— Hydro science & Engineering, C. Maxwell Stanley Hydraulics Laboratory, The University of Iowa.

André Bakker. Applied Computational Fluid Dynamics

Metin Ozen, Ph.D., CFD Research Corporation Ashok Das, Ph.D., Applied Materials Kim Parnell, Ph.D., Parnell Engineering and Consulting. *CFD Fundamentals and Applications.*

Introduction to Computational Fluid Dynamics (CFD)

Wenbin Song, Andy Keane, Hakki Eres, Graeme Pound, and Simon Cox. *Two Dimensional Airfoil Optimization Using CFD in a Grid Computing Environment.*

Leo Lazauskas, Cyberiad, First Draft: 16 Dec. 2009, Resistance and Squat of Surface Combatant DTMB Model 5415: Experiments and Predictions

Dimitri J. Mavriplis, ICASE, NASA Langley Research Center, Hampton, VA 23681, USA (September 15-18, 2002), Unstructured Mesh Related Issues In Computational Fluid Dynamics (CFD) – Based Analysis And Design. (*11th International Meshing Roundtable, Ithaca New York, USA*)

Multiphase Flow Modeling. CFX solver tutorial

C. Böhm, R&D Centre Univ. Appl. Sciences Kiel, Yacht Research Unit, Germany **K. Graf**, Univ. Applied Sciences Kiel, Germany Validation Of Ranse Simulations Of A Fully Appended Acc V5 Design Using Towing Tank Data.

Gianpiero Lavini¹, Lorenzo Pedone¹, Davide Harpo Genuzio Application of fully viscous CFD codes in the design of non propellers for passenger vessels

B. Godderidge¹ A.B. Phillips, S. Lewis, S.R. Turnock, D.A. Hudson, and M. Tan. Fluid-Structure Interactions Research Group, Froude Building, School of Engineering Sciences, University of Southampton, SO17 1BJ, UK. The simulation of free surface flows with Computational Fluid Dynamics

Madhusuden Agrawal, ANSYS. Multiphase flow modeling in ANSYS CFD

Khairul Hassan, Maurice F. White. *CFD Applications in Ship Design Optimization*

Jean Jacques Maisonneuvz, Frédéric Dauce, Bertrand Alessandrini. *Towards Ship Optimal Design Involving CFD.*