Chapter 4
Simulation of Ship on Head Waves using CFD Analysis

Lilly.R *, Prabhakaran.S, Jayasurya.R, Yarabala Gopi Krishna

AMET University, Chennai - 603112, India.

** Corresponding Author: lilly1979@ametuniv.ac.in*

Abstract

For modelling and analysing fluid transport events, computational fluid dynamics (CFD) is one of the most crucial techniques. It is frequently applied to understand flow patterns and dynamic effects brought on by fluid motions in a variety of engineering fields. Over the past two decades, CFD has made considerable strides in the field of maritime technology, particularly in the area of ship hydrodynamics. For a Panamax bulk carrier measuring 217 metres in length, HSVA conducted model experiments in 2009 for a variety of Froude values and head waves with two distinct wave durations. Significantly less than the ship length were the two wavelengths that were tested. As a result, it is reasonable to assume that the ship motions are negligible. Accordingly, the majority of the additional wave resistance is brought on by wave reflections (diffraction) off the ship's bow. Two Froude numbers were simulated for both head waves and calm water. The calculated results for both simulated Froude numbers in calm water and in waves demonstrate a good agreement with the data. For the calm water condition at the lowest Froude Number, the maximum deviation from the measurement is less than 5%.

Keywords: Computational Fluid Dynamics, STAR CCM+, Head waves, Hull pressure, Wave reflections.

1. Introduction

Using approximations that solve partial differential equations (PDEs) regulating flows, CFD has been used to predict how fluid flow will behave. The process for assessing how precisely the CFD model represents the real environment from the

ISBN 978-819620981-0

perspective of the intended application is essential to the model's success. To be confident in the predicted outcomes, CFD solutions must show the error bounds and uncertainties involved in the findings [1].

Computational Fluid Dynamics (CFD) is a method for modelling, predicting, and visualising how fluids, such as gas or liquid, flow using techniques from physics, applied mathematics, and computer science [2]. With the use of CFD, which employs mathematical modelling tools, numerical computing, and software tools to create, comprehend, construct, and so anticipate the required scenarios, a qualitative and quantitative prediction can be made. When compared to physical modelling techniques for complicated modelling geometries, CFD gives faster results since physical modelling techniques demand more time, space, and money [3]. CFD enables for a more systematic and visual examination of in-depth queries. This aids in the development of new theoretical innovations in a variety of domains for future advancements [4]. CFD is also a cost-effective method since it converts actual fluids into digital pictures, allowing for a more thorough, longer, and uncorrupted examination. This is demonstrated by breakthroughs in numerous fields and the resulting cost reductions. CFD is also used to produce controlled simulations that aren't based on real-world scenarios [5-7].

Modelling physical processes is an activity that involves analysing, evaluating, simulating, and defining common knowledge in a way that people can understand. Its main goal is to use simulations to deliver crystal clear information about the object and to convey a conceptual knowledge of the physical model [8-9]. All models are based on simulacra, or consideration of reality, and they are far more valuable than empirical object. For recurrent processes, CFD modelling is an iterative process with no definite answers. Because the mathematical equations employed in CFD are not linear, the variable may have a variety of values, creating convergence in turbulence modelling [10-12]. Convergence can be accomplished by defining a degree of accuracy that is close to the desired results. It's now just a matter of deciding which criterion to apply to keep convergence to a minimum [13].

ISBN 978-819620981-0
9 788196 209810

The numerical and mathematical formulas are not included in this CFD approach. In this section, the numerical schemes of the STAR-CCM+ are provided, and special CFD modelling considerations for completely nonlinear sea-keeping analysis are covered [14-15]. Because of the trimaran's numerous benefits, extensive research has been conducted in recent years, including studies on its hydrodynamic performance. The majority of hydrodynamics research centred on optimising side hull placement to get the best possible interference between hulls and lowest resistance [16-17]. The ability to assess the behaviour of such vessels in waves, however, becomes increasingly demanding as the number of high-speed trimarans operating globally increases. Model testing and numerical prediction are typically used methods to look into seakeeping properties [18]. In this paper, research on CFD computation of ship motions and additional resistance in waves for a high-speed trimaran is conducted by solving RANSE (Reynolds Averaged Navier-Stokes Equations). By using the finite volume method, the governing equations are discretized. To deal with the nonlinear free surface, the volume of fluid approach is used. The estimated findings of ship motion and additional resistance for a high-speed trimaran are reported in this study. The experiment's findings on seakeeping for high [11-12].

2. Hull Surface Generation

The numerical method described in this study allows for accurate prediction of wave-induced motions and additional resistance for trimarans moving through normal head waves. The majority of discrepancies between numerical and experimental results are tolerable. By taking into account all the inertial and hydrodynamic forces and moments acting on the rigid body, the equations for ship motion in waves are produced. Geometric particulars of ship for the hull surface generation is given in the Table 1.

ISBN 978-819620981-0

Table 1: Geometric particulars of ship

S.No	Details	Nomenclature	Value with units
1	Length between perpendiculars	LBP	203 m
2	Breadth	B	33 m
3	Depth	D	19.5 m
4	Draft	T	13.6 m
5	Speed	V	13.5knots

3. Lines Plan

A lines plan is a specific drawing that is used to depict a ship's whole hull form. The plan view is the first of three views in a lines plan. It explains the vessel's top-down perspective, with waterlines, the profile view, which shows the ship's side view, with buttock lines, and the body plan view, which shows the vessel's ends, with station lines. The lines plan of the geometry of the ship is shown in Figure 1. The 3D modelling was done for the bulk carrier and it is shown in the Figure 2.

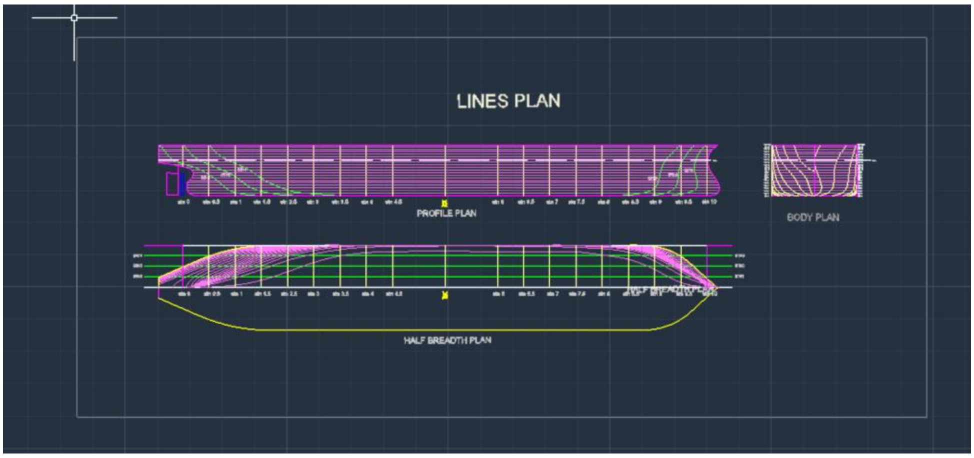

Figure 1: Lines plan of the Ship geometry

ISBN 978-819620981-0
9 788196 209810

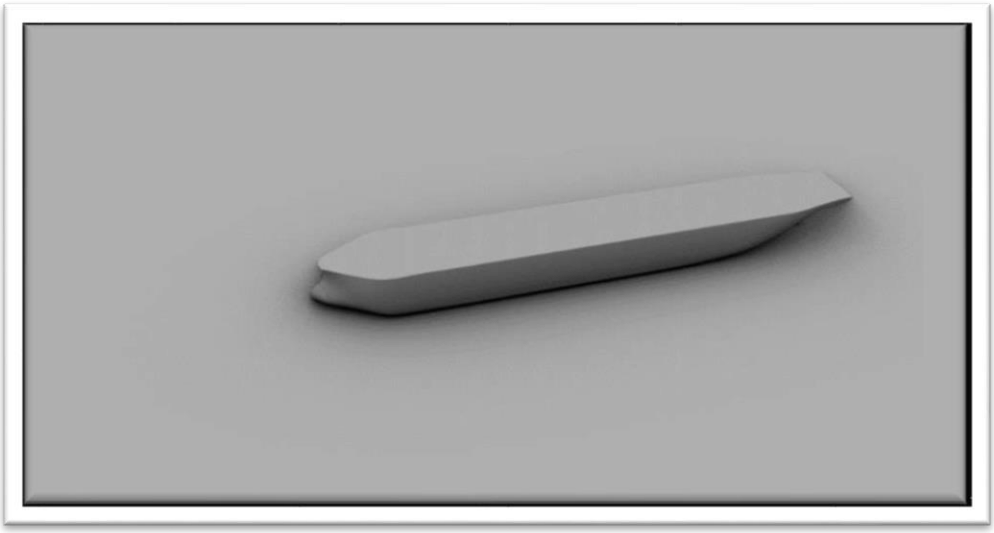

Figure 2: 3D model of bulk carrier

4. Methodology

4.1 STAR-CCM+

Simcenter STAR-CCM+ is a Multiphysics computational fluid dynamics (CFD) tool for simulating the performance of products in actual environments. Every engineer's CFD simulation toolkit now includes automatic design exploration and optimization thanks to Simcenter STAR-CCM+. Everything from CAD, automatic meshing, Multiphysics CFD, complex post processing, and design exploration are all included in the single integrated platform. Engineers may efficiently explore the whole design space as a result, leading to quicker and better design decisions [19].

4.2 Mesh Scene and Scalar Scene

Meshing is the process of dividing the continuous geometric space of an object into tens of thousands or more different shapes in order to accurately define the physical shape of the thing. High-fidelity simulations are made possible by 3D CAD models that are more accurate the more complex a mesh is. It is the responsibility of the mesh generator to divide the flow domain into control volumes. The location of the nodes in relation to the vertices depends, for instance, on whether the solver uses cell-

ISBN 978-819620981-0

centered or cell-vertex storage. The complexity increases if a velocity grid with staggered steps is employed. The meshing was done for the ship and it is shown in the Figure 3.The scalar scene explains the distribution of the parameters for the part of interest. It is done for the ship and it is shown in the Figure 4.

4.3 Simulation

The results of the simulation was carried out with the iteration steps of 1700 and the model is runned for a period of 40 minutes .The iteration process was continued upto the convergence takes place in the model. It is better studied with the study of residuals. It shows the difference in between the last iteration and the current one. Each solved solution and each mesh cell will have a unique set of residuals. Each CFD code's method for normalising the solution residuals will be distinct. They will never approach exactly zero in any numerical method; but, the smaller they are, the more likely the simulation has reached its convergence. The residual graph in Figure 5 demonstrates that for all of the tested conservation equations, the residuals reached relatively low and constant values around the 900th iteration, indicating that convergence was likely reached.

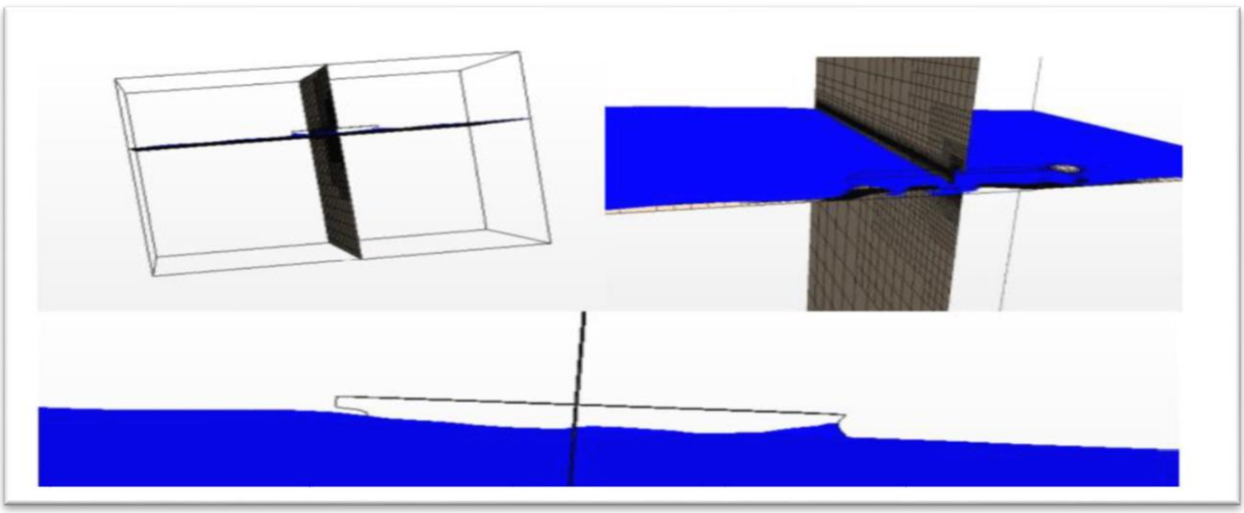

Figure 3: Three Dimensional Meshing Scene

ISBN 978-819620981-0

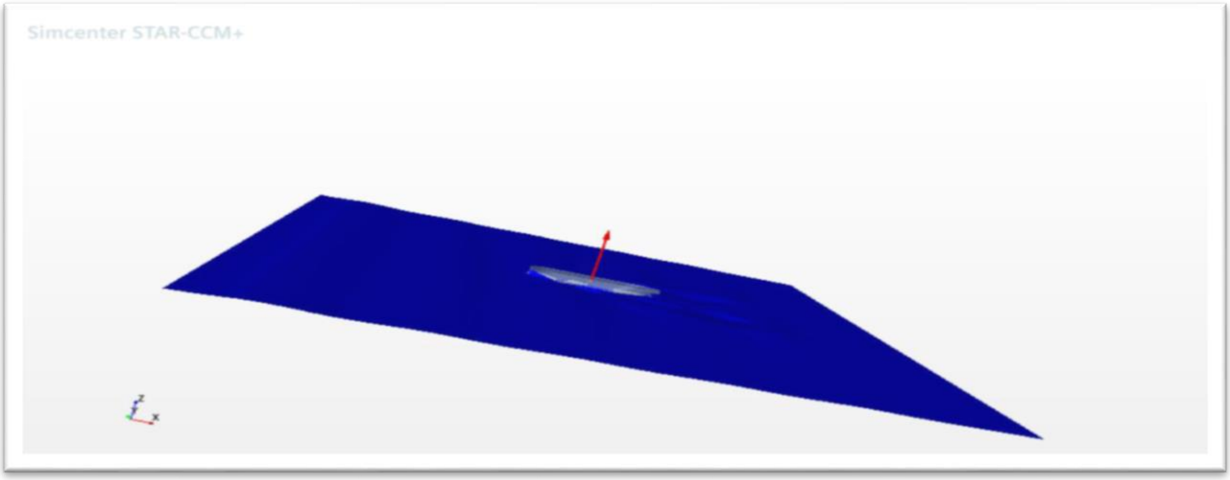

Figure 4: Three Dimensional Scalar Scene

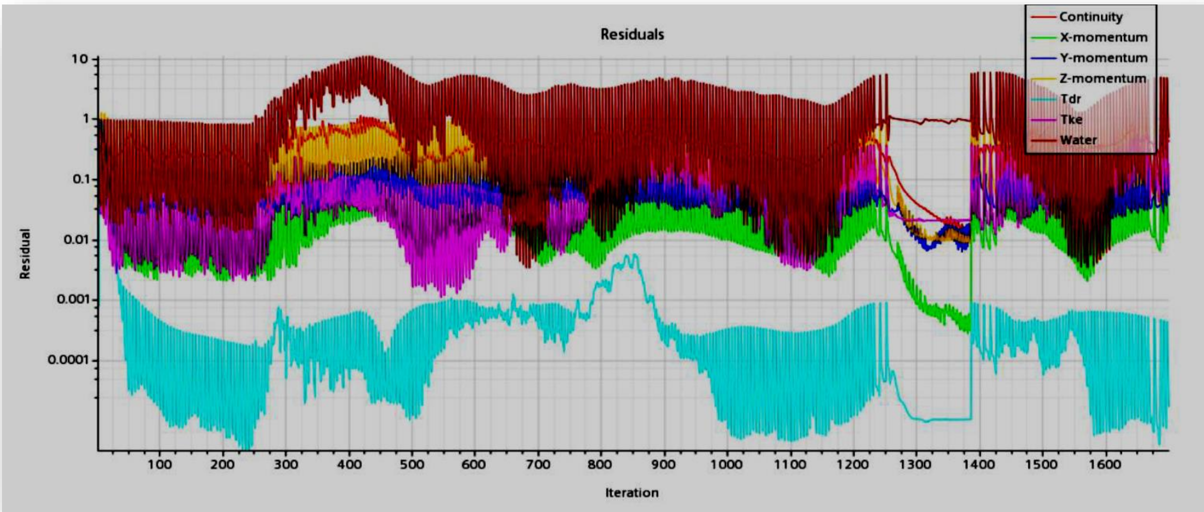

Figure 5: Residual Graph

5. Analysis and Discussion

5.1 Hull pressure

A submarine's inner hull is designed to withstand pressure while submerged. The most widely acknowledged method for estimating hull pressure is to divide the reaction of the marine fender over the whole frontal flat area of the marine fender panel to get the average hull pressure. The hull pressure simulated for 1700 iterations

ISBN 978-819620981-0

is given in the Figure 6.The pressure predicted in the hull founds to be in the range of 12.206 Pa to 3574 Pa in the bottom of the hull.

5.2 Parameter plots

The convergence of a simulation can be well understood by creating a plot. The evolution of the selected parameters with the progress of iterations .Once the simulation is over, the correspondent value will be shown in the output window. The report made with the interesting magnitudes of the parameters selected .In this simulation the report was generated Y rotation monitor plot with the proper interval of physical time 0.2sec and the convergence seems to be occurred constant after 2.4 sec and it is shown in the Figure 7.

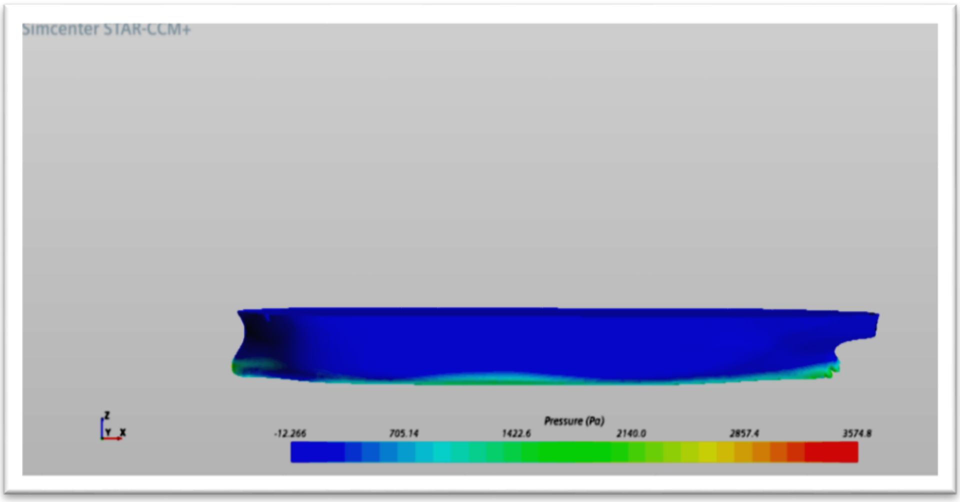

Figure 6: Hull pressure

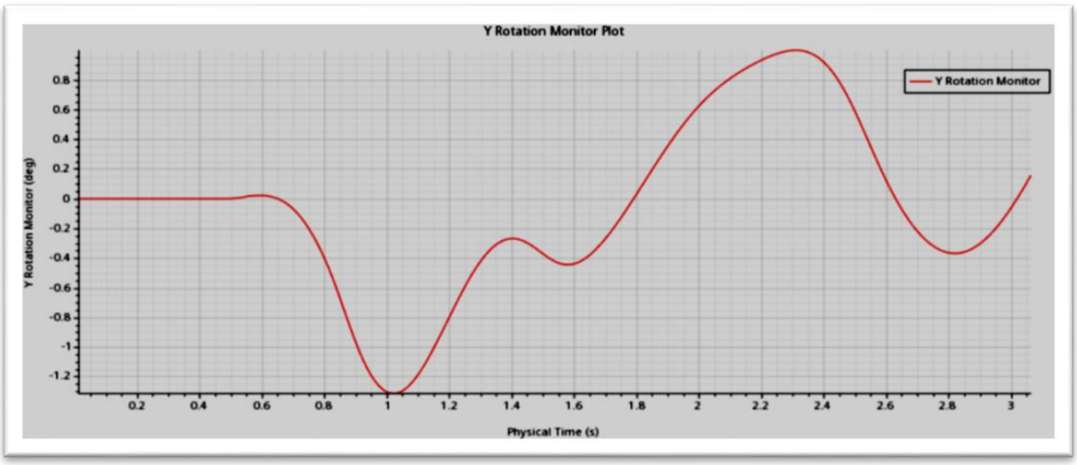

Figure 7: Y Rotation Monitor plot

ISBN 978-819620981-0

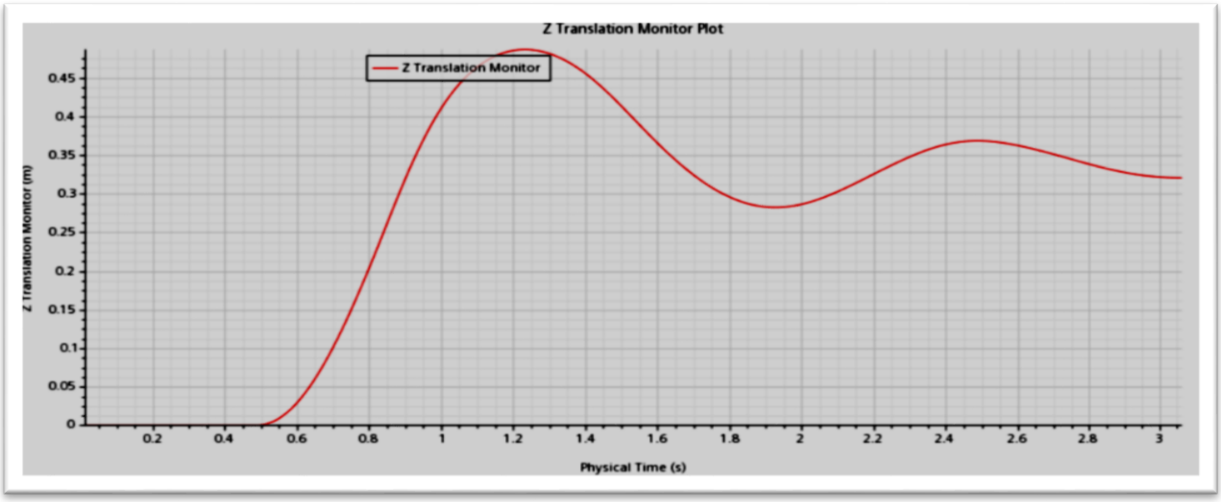

Figure 8: Y Rotation Monitor plot

The simulation report for Z translation monitor shows the convergence founds to be constant after 0.8sec and it is shown in the Figure 8. The output window shows the values of the iteration and the values are very agreeable to both the conditions of the iterations .The results are shown in the Figure 9.

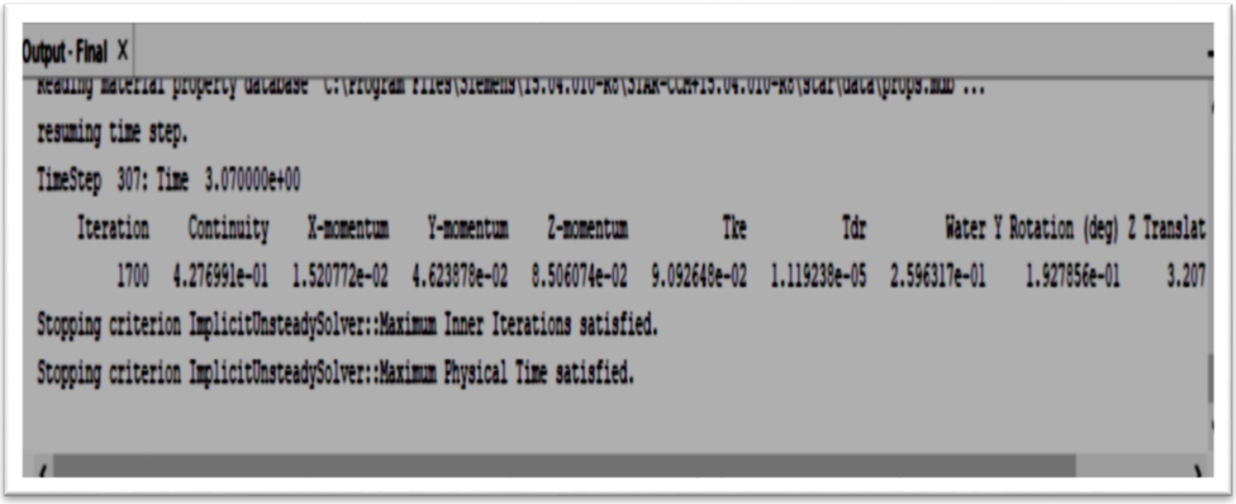

Figure 9: Output of the Simulation

6. Conclusion

Time integration techniques, such as a second-order predictor-corrector scheme, are to be used to solve the 6-DOF equations of motion. Since there is no hydrostatic restoring mechanism for the surge sway-yaw motions in the horizontal plane, a numerical course-keeping model would be necessary to prevent irrational drift

ISBN 978-819620981-0

motions throughout the motion simulation. Otherwise, a simpler approach might merely be seen of as the heavy-roll-pitch motions in vertical plane. Ship appendages like bilge keels and rudders must be taken into account in the geometry modelling for correct roll damping modelling. In the alternative, the roll motion simulation may be performed using a streamlined roll-damping model. Additionally, a ship's forward position is shown via a numerical wave probe. Using this wave probe, you can keep an eye on the wave heights during the CFD simulation. A very slight drop in wave heights was seen for the current mesh size and time step (just approximately 2% each wave cycle), indicating very low numerical level.

References

[1] American Bureau of Shipping, "Guide for Safe Hull Dynamic Loading approach" for Vessels, ABS,2006.

[2] Chen, H.C. and Yu, K., "CFD simulation of wave current-body interactions including green water and wet deck slamming", Journal of Computers and Fluids, Vol.38, No. 5, pp. 970-980,2009.

[3] Draper, Norman R., and Box, George E.P., "Empirical model-building and response surfaces ", John Wiley and sons,1987.

[4] El Hagrasy, A.S., Hennenkamp, J.R., Burke, M.D., Cartwright, J.J., and Litster, J.D., "Twin screw wet granulation: Influence of formulation parameters on granule properties and growth behaviour", Powder Technology ,2013.

[5] Ferziger, J.H. and Peric, M.,"Computational Methods for Fluid Dynamics", 3rd ed.,Springer, Berlin, Germany,2003.

[6] Guang Xu, Kray D. Luxbacher, SaadRagab, JialinXu, and Xuhan Ding, "Computational fluid dynamics applied to mining engineering: a review", International Journal of Mining, Reclamation and Environment, Vol. 31, Iss. 4, pp. 251-275,2017.

[7] Hu, C. Kashiwagi, M.,"Validation of CIP-based method for strongly nonlinear wave-body interactions", 26th Symposium on Naval Hydrodynamics, Rome, Italy,2006.

[8] Kim, S. Shin, Y. and Liu, D.,"Advanced dynamic loading approach for large container carriers based on nonlinear seakeeping analysis", ISOPE conference, San Francisco, USA,2006.

[9] Kim, S. Yu, H.C. and Hong, S.Y., "Segmented model testing and numerical analysis of wave-induced extreme and springing loads on large container carriers", ISOPE Conference, Beijing, China,2010.

ISBN 978-819620981-0

[10] Linfield, K. W., and Mudry, R. G.,"Pros and cons of CFD and physical flow modelling",2008.

[11] Praveen kumar.R, "CFD Analysis of Pelton Bucket", International Journal of Mechanical and Production Engineering Research and Development,Vol 8,Issue 1,pp 775-780.

[12] Raase, Sebastian, and Nordström, Tomas,"On the Use of a Many-core Processor for Computational Fluid Dynamics Simulations", Procedia Computer Science, Vol. 51,2015.

[13] Raju M.S.P., Sivabalan P., Thamby T., Saravanan B., "Effect of bulbous bow on resistance of a tuna longliner", International Journal of Advanced Research in Engineering and Technology,Vol 11,Issue 2,pp 136 -145,2018.

[14] Shameem B. M, "CFD analysis and experimental validation on the effectiveness of bilge keel as a roll stabilizer", Journal of Engineering and Applied Sciences,Vol 13,pp 9403-9407,2018.

[15] Thabit, Thabit H., and Younus, Saif Q.,"Risk Assessment and Management in Construction Industries", International Journal of Research and Engineering, Vol. 5, No. 2, pp. 315-320,2018.

[16] The Seakeeping Committee,"Final report and recommendations to the 25th ITTC", Proceedings of the 25th ITTC, Vol. 1, Fukuoka, Japan,2008.

[17] Tu, J., Yeoh, G., and Liu, C., "Computational Fluid Dynamics – A practical Approach", 2nd Edition, Elsevier, UK,2012.

[18] Yang, C. Lu, H. L¨ohner, R. Liang, X. and Yang, J.,"Numerical Simulation of Highly Nonlinear Wave-Body Interactions with Experimental Validation", Proc International Conference Violent Flows, VF-2007, Fukuoka, Japan,2007.

[19] Yu, H.C. Kim, S. Yook, R.-H. Lee, D.-Y. and Jung, H.-C., "The Effect of Icebreaking Bow on the Open Water Performance of a Large Arctic Ore Carrier", ICETECH, Alaska, USA,2010.

https://doi.org/10.5281/zenodo.7657560

SCIENTIFIC RESEARCH REPORTS

www.srrbooks.in

ISBN 978-819620981-0