Acta Universitatis Danubius

ŒCONOMICA

The Use of CFD Methods in the Shipbuilding Industry and their Benefits

Ana-Maria Chirosca[1], Liliana Rusu[2]

Abstract: CFD (Computational Fluid Dynamics) is a branch of fluid mechanics that uses numerical methods and algorithms to analyze and solve problems involving fluid flow. In general, in the shipbuilding industry, when it is desired to validate a ship model, in order to determine its resistance, as well as the ship's behavior on waves, model tests are used in towing tanks. But resistance in calm waters and resistance in waves are not the only naval characteristics studied in the towing tanks. There are also tests in the basin and cavitation tunnels for testing propellers, tests for vessel's maneuvering behavior, and much more. Unfortunately, these tests involve high prices and last a long time, and in the future the aim is to replace these tests with CFD simulations that last much less and involve much lower costs, and the accuracy of the results is satisfactory. This study provides an overview of CFD simulations, their benefits and disadvantages.

Keywords: CFD simulation; ship industry; towing tanks; ship design

JEL Classification:

1. Introduction

CFD comes from Computational Fluid Dynamic and is a branch of fluid mechanics that studies the behavior of fluids and physical systems, through computer simulations performed on the computer, instead of the classic use of physical tests.

[1] PhD in progress Eng., Faculty of Engineering, Department Mechanical Engineering, “Dunarea de Jos” University of Galati, Romania, Address: 47 Domneasca Street, Galati, Romania, Corresponding author: oanacenac@yahoo.com.

[2] PhD, Faculty of Engineering, Department Mechanical Engineering, “Dunarea de Jos” University of Galati, Romania, Address: 47 Domneasca Street, Galati, Romania.

CFD methods can be used for a wide range of problems in many fields of research (Zhang, et al., 2006; Wei, 2017, The development and application of CFD technology in mechanical engineering), and the main fields are aerospace, aerodynamics, biological engineering, heat transfer, fluid flow, weather simulation, and the field of interest for this paper, naval engineering.

The shipbuilding industry is evolving at the same time as technology, and because of this the latest methods in the shipbuilding industry must be correlated with the development of technology and implemented in order to optimize and verify the performance of ships as quickly and efficiently as possible.

Ship hydrodynamics is a complex and complicated problem and solving these problems by conventional methods involves a costly and time-consuming experiment in towing tank. Although CFD simulations are not a new concept, they could be applied to solve hydrodynamic problems with improved computer and processing performance, which currently allow results to be obtained in a short time and at a much lower total price.

This paper includes a presentation of the problems solved by numerical simulations using CFD Methods in the shipbuilding industry and the software that can be used. It also presents the steps to be followed to perform such a simulation, as well as the benefits and disadvantages of using these increasingly popular methods.

2. CFD in Practical Ship Design

When it comes to ship design, the hydrodynamic characteristics of the ship must be studied and evaluated from the initial design to continue with the next design steps. The main parameters that are taken into account in the initial stage and the subject of many studies regarding CFD simulations are the resistance of the ship and the propulsion power (A.-M Chirosca, L. Rusu, 2021, Comparison between Model Test and Three CFD Studies for a Benchmark Container Ship), the behavior of the ship in waves (A.-M Chirosca, L. Rusu, F. Pacuraru, 2021, Study on the behavior of benchmark container ships in regular waves), the manoeuvrability of the ship (Ravindra, Kudupudi, Saji, Das & Nirjhar, 2014) and the optimization of the ship hull form (Aksenov, Pechenyuk & Vučinić, 2015).

CFD methods not only allow a much faster evaluation of these characteristics compared to towing tests, but also allow a much faster optimization of the ship's hull, a step that would involve the manufacture of a new model and its retesting in the towing tank, if the classical method is used.

Using these methods requires a high-performance computer with as many processors as possible, as well as paying for a commercial software license, which is quite expensive.

In a design process, the main hydrodynamic problems are the ship resistance in calm water and the self-propulsion characteristics, and in general, the first numerical simulation consists in checking the calm water resistance at the design speed.

In order to study the behavior of the ship, simulations are performed on regular waves and on irregular waves, but unfortunately, this type of simulation requires more computation time.

In addition to these types of general simulations that need to be performed, other, more complex problems can be solved, such as maneuvering, sloshing, slamming, green water events, and more.

To solve the problems of ship design and naval engineering, there is a range of software that we can use, and the best known are: FINE Marine, SHIPFLOW, ANSYS FLUENT CFD, ANSYS CFX, OpenFOAM, Star CCM +.

FINE Marine is the most widely used software in the shipbuilding industry, due to the fact that it is a software dedicated to ship hydrodynamics and can simulate the flow around any type of ship, from containers, tanks, to yachts, hydrofoils and even propellers.

SHIPFLOW is also a dedicated ship design software, which can simulate the flow around a ship, determine the resistance and propulsion characteristics of the hull, optimize the ship and study the performance of seakeeping.

Compared to FineMarine and SHIPFLOW, Ansys is a general-purpose software that deals with a variety of mechanical issues. It has implemented two modules that are used in the shipping industry, Ansys Fluent and Ansys CFX, which offer highly accurate solutions.

Related to Ansys, OpenFOAM and STAR-CCM+ are also general-purpose software that can solve the ship's hydrodynamic problems, but the great advantage of using OpenFOAM is that it is publicly available software.

3. Typical Workflow of CFD Simulation

Performing a CFD simulation can be divided into three main stages (Abhiroop, Varada & Shameem, 2018), which consist of the pre-processing stage of the data of the problem to be analyzed, the simulation itself, and the last step, the analysis and visualization of the simulation results. The workflow of a CFD simulation is presented in Figure 1.

The first stage, pre-processing, involves several intermediate steps, such as importing the geometry or generating the geometry, generating the mesh, and setting up the case.

The geometry should be carefully checked for missing faces, surface gaps, overlapping faces, sharp angles, and details that are not useful in simulation should be cleared.

Mesh generation (Seo et all, 2010) is the most important step in a CFD simulation in the field of flow simulations, and if it does not meet the accuracy limit and does not capture all flow features it can alter the expected results. This step involves dividing the physical domain into several regions called control cells or volumes.

The accuracy of the discretization grid depends on the size of the cells and the total number of cells in the range defined for the simulation, this aspect being limited by the capabilities of the computer on which we perform the simulation. The quality of the discretization grid can be improved by increasing the degree of fineness only in the areas of interest, by checking the aspect ratio and the shewness (ITTC, 2011), checking the number of concave cells, convex cells and twisted cells, as well as other factors.

The last step in the pre-processing stage is the configuration of the case, which defines the material properties, initial conditions and boundary conditions, physical model, numerical schemes and stabilization options, run time parameters and case parameterization for optimization simulations.

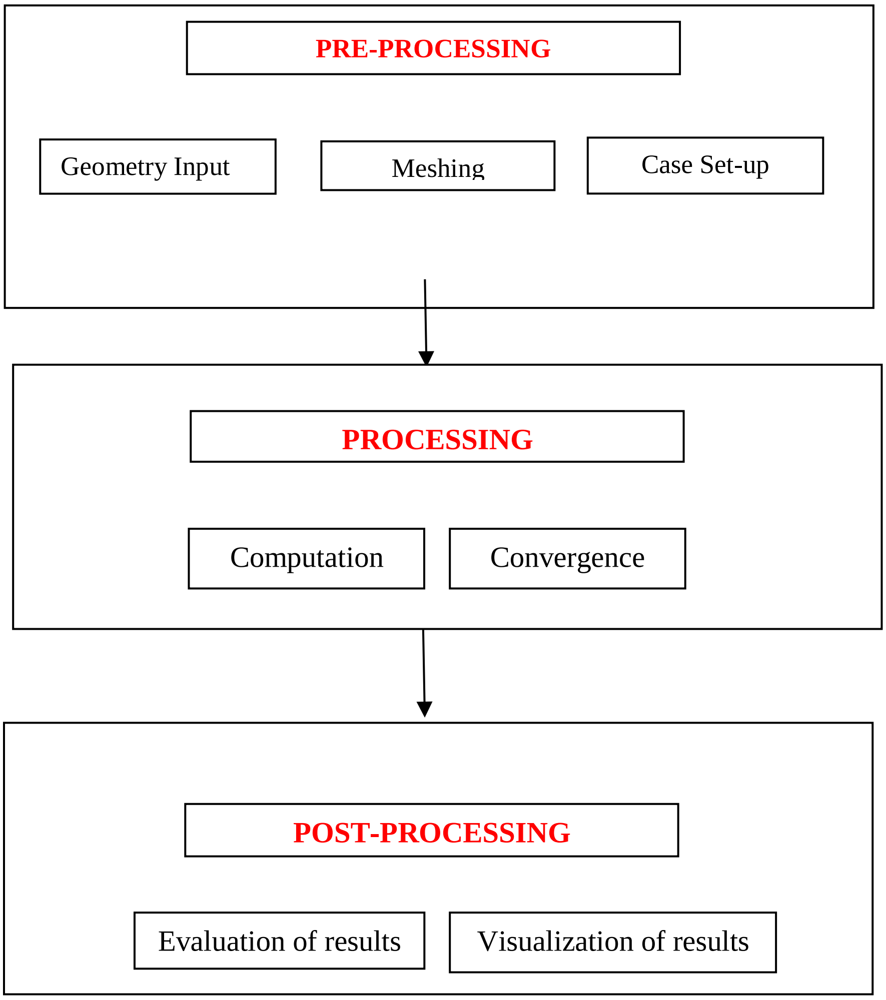

Figure 1. Typical Workflow for a CFD Simulation

After establishing all the necessary parameters for the case, the second step is to launch the simulation and monitor the achievement of the convergence of the solution.

The last and most important stage, called post-processing, is the analysis of the results of the computation. At this stage, the results are verified and visualized using various methods, such as contour diagrams, vector diagrams, streamlines, and design optimization decisions can be made based on the results obtained.

The steps presented are the general steps required to perform a CFD simulation from scratch. Certainly, some steps may be missing, for example by directly importing the mesh for the domain.

4. Benefits and Disadvantages of CFD Methods

The short time in which solutions can be obtained is the most important advantage of CFD simulation.

The major advantages (W. Kevin et al., 2008, Pros and Cons of CFD and Physical Flow Modeling, August,) over physical fluid dynamics experiments are that flow conditions that cannot be reproduced experimentally can be simulated using CFD simulations, and much clearer and more detailed information about the model can be studied. Also, another advantage is that the size of the domain can be scaled and thus simulations can be computed both for the full-scale and the model scale.

The control of the set-up configuration makes it possible to run and manage multiple simulations and so the optimization process can be done quickly.

Although the benefits of using CFD simulations instead of physical tests in towing tanks or cavitation tunnel, they also have several disadvantages. The main disadvantage is the cost of a license for using a CFD software, which, unfortunately, is quite high. However, the overall price is lower using CFD Methods instead of physical tests.

Also, the need for a high-performance computer is a disadvantage, the time required for the simulation, as well as the size of the discretization of the domain are limited by the characteristics of the computer on which the simulation is performed.

Last but not least, although the steps required for a simulation are known, and CFD software has a friendly interface, more complex problems require an experienced person who understands the phenomenon itself, as well as the theory behind the program and can simulate the problem as realistically as possible.

5. Conclusion

CFD methods have become a popular option and solution, especially due to the difficulties encountered during experimental testing, but also the much lower price.

In the shipbuilding industry, CFD methods can be used to study ship performance under realistic operating conditions and determine hull resistance, propeller performance prediction, including cavitation, ship wave behavior, maneuverability, and other issues.

The workflow for CFD simulations consists of three main parts. The pre-processing is the first step and consists in creating geometry, discretizing the domain and configuring the parameters. The second stage is represented by the computation, and the last stage, post-processing, which consists in visualizing and verifying the obtained solutions.

By parameterizing the configuration, many more tests can be performed, in a much shorter time, compared to the generation of the model and their experimental testing.

As shown in many studies (Chirosca & Rusu, 2021), the accuracy of the results between numerical simulations and experiential testing is satisfactory, and the difference is in the range of 2% to 5%.

Thus, CFD simulations have a wide range of advantages, but attention must be paid to the series of limitations, depending on the performance of the computer used, as well as the experience in understanding the phenomena to be simulated.

Acknowledgement

"The work of the first author is supported by the project ANTREPRENORDOC, in the framework of Human Resources Development Operational Programme 2014-2020, financed from the European Social Fund under the contract number 36355/23.05.2019 HRD OP /380/6/13 – SMIS Code: 123847. The work of the second author was carried out in the framework of the research project DREAM (Dynamics of the REsources and technological Advance in harvesting Marine renewable energy), supported by the Romanian Executive Agency for Higher Education, Research, Development and Innovation Funding – UEFISCDI, grant number PN-III-P4-ID-PCE-2020-0008."

References

*** ITTC – Recommended Procedures and Guidelines, Practical Guidelines for Ship CFD Applications. (2011). *26th ITTC Specialist Committee on CFD in Marine Hydrodynamics*

Abhiroop, K.; Saidas, Varada & Shameem, B. M. (2018). A review on the advancement of CFD technique in ship hydrodynamics. *International Journal of Computational Engineering Research (IJCER)*, Volume 08, Issue 12.

Aksenov, A. A.; Pechenyuk, A.V. & Vučinić. D. (2015). Ship hull form design and optimization based on CFD. *Taylor & Francis Group*, London.

Chirosca, A.-M & Rusu, L. (2021). Comparison between Model Test and Three CFD Studies for a Benchmark Container Ship. *J. Mar. Sci. Eng.*, 9, 62.

Chirosca, A.-M; Rusu, L. & Pacuraru, F. (2021). Study on the behavior of benchmark container ships in regular waves, *IOP Conf. Ser.: Mater. Sci. Eng*., 1182 012013.

Kudupudi, Babu, Ravindra & Saji, V. & Das, H. Nirjhar (2014). CFD simulation of ship maneuvering. *Conference:International Conference on Computational Experimental Marine Hydrodynamics MARHY*.

Linfield, Kevin W.; Robert G. P.E.& Mudry, P. E. (2008). Pros and Cons of CFD and Physical Flow Modeling, August. *Airflow Sciences Corporation*.

Seo, Jeong Hwa; Seol, Dong Myung; Lee, Ju Hyun; Rhee, Shin Hyung Flexible (2010). CFD meshing strategy for prediction of ship resistance and propulsion performance, *International Journal of Naval Architecture and Ocean Engineering*, Volume 2, Issue 3, pp. 139-145.

Wei, Yufeng (2017). The development and application of CFD technology in mechanical engineering. *IOP Conf. Series: Materials Science and Engineering*.

Zhi-rong, Zhang; Hui, Liu; Song-ping, Zhu & Feng Zhao (2016). Application of CFD in Ship Engineering design practice and Ship Hydrodynamics. *Journal of Hydrodynamics*, Ser. B, Volume 18, Issue 3, Supplement, pp. 315-322.