METHOD OF OUTFITTING DESIGN FOR EFFICIENT DRAINAGE IN BALLAST TANK

劉, 廣帥

https://hdl.handle.net/2324/4784564

出版情報：Kyushu University, 2021, 博士（工学）, 課程博士
バージョン：

（様式２）

氏　　名：劉　廣帥（リュウ　グアンシュアイ）

論 文 名：METHOD OF OUTFITTING DESIGN FOR EFFICIENT DRAINAGE IN BALLAST TANK

（艤装設計におけるバラストタンク浚水の効率化に関する研究）

区　　分：甲

論　文　内　容　の　要　旨

This paper presents the analysis of ballast tank drainage efficiency using hydraulic experiments and CFD (Computational Fluid Dynamics) simulations.

The ship ballast system provides enough seaworthiness conditions for large cargo ships in terms of stability, trim, strength, and propulsion efficiency when sailing with empty tanks. In some special loading conditions, the ballast system is also responsible for adjusting the ship's navigation posture and compensating for stability.

At present, due to the high concern of environmental protection topics, the research on ballast water mainly focuses on water pollution, including ballast water exchange, ballast water treatment, and so on. However, there are few studies on the issue of ballast water drainage efficiency during ship loading. The improvement of ballast water discharge efficiency can save time for ship loading, thereby reducing berth rental costs. It can also reduce the waste of ship loadings and provide additional economic value for shipping companies. In addition, the arrangement of the ballast system is closely related to the structure of the ship's ballast tanks, and a reasonable arrangement can also help improve the strength of the hull.

The design of the ballast system is a system engineering problem, which requires comprehensive consideration of various factors such as pumping rate, bell-mouth location, drain hole size and arrangement, structure strength, equipment cost, etc. The ballast tank is divided into many compartments by structural members such as longitudinals, transverses, and side girders, and the compartments are connected by lightening holes, drainage holes, etc. The structure is complicated. Therefore, at present, the design of the ballast system is mostly based on previous experience, and there is no mature design method that can evaluate and predict the discharge efficiency of ballast systems.

The purpose of this research is to establish an experimentally validated computational model for studying the flow phenomena in ballast tanks and evaluating the discharge efficiency to provide a basis for the design and optimization of ballast tanks. The study takes a ballast tank of a typical bulk carrier as the research object, and a 1:10 scale experimental model is made to collect experimental data. Then numerical simulations are performed by the CFD method and compared with the experimental results to verify the credibility of the numerical simulation results. For the simulated environment where two phases of air and water coexist in the ballast tank, the VOF (volume of fluid) method is utilized in this paper to trace the interface between the two phases to obtain the changing state of the water surface during the ballast water discharge.

At the beginning of the study, the simulation of the single longitudinal and single transverse configuration ballast tank model with six compartments are completed, and the results obtained are in good agreement with the experimental results, which initially verified the feasibility of the VOF method in the simple ballast tank model drainage simulation. Then, this study extends the simulation object to the complex two-dimensional configuration ballast tank model, and the simulation with the 4x4 compartments model is completed. The results are also in good agreement with the experiment. After that, in the ballast tank model with the 8x8 compartments model, the comparative study of experiment and numerical simulation is completed in accordance with the drainage hole configuration plan of the actual ship. From analyzing the numerical simulation and the experimental results, some flow phenomena and laws are observed. Then, some improvements to the configuration of drain holes in the ballast tanks of real ships have been proposed, which are drain courses. From the experimental comparison, the improvement schemes do help to improve the drainage efficiency of the ballast tank.

Because the CFD simulation requires many computing resources, the supercomputer system in Kyushu University is used for calculations. But this is not favorable for the engineering evaluation of ballast systems. Therefore, we introduced a way to construct a simplified mathematical model to assist in evaluation and design. This simplified mathematical model has been initially validated in the single longitudinal and single transverse configuration ballast tank models and the 4x4 model.

The thesis is composed of seven chapters and the outline is described as follows:

Chapter 1 briefly introduces the research topic, the existing ballast tank drainage overview, and explains the complexity of ballast system design. This chapter also describes the motivation, objectives, and main contributions in publications throughout this research.

Chapter 2 describes the operation process of the hydraulic experiments in detail. It is mainly divided into two parts. The first part introduces the experimental equipment, the model making, the operation, and the data processing methods. The second part mainly introduces the analysis of experimental data and the key points to be observed.

Chapter 3 mainly introduces the basic equations and methods of numerical simulation. As well as the advantages and problems of numerical simulation, the necessity of using supercomputers for calculation is proposed. The processing method of computational data is also given.

Chapter 4 focuses on the comparative analysis of experimental results and simulation results. It includes the single longitudinal and single transverse configuration ballast tank model, the complex ballast tank model with a two-dimensional arrangement, and a model based on the ballast tank design scheme of a real ship. Based on the comparison, the reliability of the VOF method in ballast tank drainage simulation is verified. Moreover, some flow phenomena and laws can be observed through the analysis of numerical simulation results, which provide references for the optimization of the ballast system and the construction of simplified mathematical models in the later section.

Chapter 5 mainly introduces the drain courses designed based on the optimization of drain hole arrangement. According to the experimental results, some constructive optimization suggestions are summarized.

Chapter 6 focuses on the construction of the simplified mathematical model. The pump influence coefficient is introduced, which makes the calculation of the method much more accurate. This method is important because it is more efficient compared to the CFD simulation.

Finally, Chapter 7 gives the conclusions of the study and recommendations for future research.