CFD applications for Ballast water management studies

Author Name(s): Ram Kumar Joga (Surveyor, IRCLASS), Deepti Poojari (Surveyor, IRCLASS), Sachin Suryakant Awasare (Surveyor-1, IRCLASS), Sharad Sham Dhavalikar (Principal Surveyor, IRCLASS), A. R. Kar (Sr. Principal Surveyor)

The method and application of Computational Fluid Dynamics (CFD) has progressed rapidly in the past fifty years in many industrial fields including marine and has been playing crucial & decisive roles. With the recent advancement in Marine CFD technology and in computational capability, practical applications of CFD in analyzing and predicting ship performance have picked up its’ pace. Analysis such as prediction of ship resistance using CFD methodology is well validated and reliable results can be obtained using CFD. Detailed flow analyses is the major advantage of CFD which otherwise is very difficult to deal in experimental techniques as well as potential based solutions. Thus the problems involving highly turbulent and non-linear flows such as green water (shipped water on decks), slamming, sloshing, etc. are effectively dealt with globally utilizing CFD methodology in recent times.

Flow patterns and flow velocities can be predicted by CFD and these parameters can be used for dealing with the flow related issues which could be critical for design and operation of vessels. Moreover, these analyses are useful not only for prediction but solution of the problem and improving the performance as well.

In the present paper simulation of Ballast tank sedimentation with CFD is discussed. Flow mixed with solid particles (e.g. mud) causes the sedimentation over the time, typically in ballast tanks causing the species to be left in tank even after drawing out ballast water. Flow analysis is performed in order to study the flow pattern and the sediment formation. Possible solution towards design changes (after observing dead zones in tanks) is suggested. Also an introduction of CFD application for ballast water treatment processes is presented.

KEY WORDS

Computational Fluid Dynamics (CFD); Ballast Water Management (BWM), International Maritime Organization (IMO), Marine Environment Protection Committee (MEPC), Ballast Water Exchange (BWE), Sedimentation, Ultra-Violet (U-V) Vibration;

INTRODUCTION

As surface ships continue to evolve and change with the demand for newer ship types, as well as newer stringent regulation requirements, the tools and techniques for design and analysis also undergo rapid changes adapting to the needs for improvements of its performance. Conventional/Empirical methods have given place to more rigorous scientific analytical and computational methods. The steady radical progress in computational methods have reduced the dependence on excessively costly tank testing and other experimental means. The two methods namely, computational and experimental are being ingeniously blended so that tank testing is minimized as the computational methods have attained maturity and reliability in hydrodynamic analysis. Therefore the term virtual towing tank is being increasingly projected as nearer to reality today, than ever before and has certainly impacted research and development in the present decade.

Computational methods are attractive because they provide better control and reduced processing time, than demanded by physical model tests. The initial application of CFD in marine field was limited to the prediction of Wave resistance which can be dealt well with conventional potential flow analysis. However, the conventional method cannot deal with complex flow problems effectively.

CFD offers many advantages that make it far advantageous to physical model testing. The CFD software can be applied to a variety of flows, including flow-related issues inside ship cavities. A key benefit of CFD is the way it provides insight into flow details. Flow quantities are computed and stored for many discrete locations in space, known as computational cells, and for many time increments, allowing the design engineer to look at arbitrary cross-sections and zoom in and out at will when post-processing the data.

In the present paper studies relevant for Ballast Water Management are discussed. With new IMO ballast water management regulations impending to curb the spread of invasive species, ballast water management systems have moved into the
spotlight for ship operators. But apart from the implementation requirements and the capital cost aspects involved, there are certain practical challenges associated with ballast water handling that can be addressed using CFD simulation.

**SEDIMENTATION**

Loading and discharging ballast water is an essential part of a ship’s operation, with large ships requiring many thousands of tonnes of water to maintain their stability, draft and maneuverability. Water from one port area is taken inside the ship and this water along with surviving organisms may be discharged at other ports based on ballast requirements. However, the species that do survive may flourish themselves in a new environment if the biological and physical conditions are favorable. Such non-native species may cause serious ecological, economic and public health impacts, particularly when they become invasive. In such cases, the solution lies in preventing them from invading or the removal of invading toxic species. Hence prevention of mixing of invasive species is an ideal solution for which various treatment methods are given in IMO Guidelines.

Following measures are suggested in MEPC 63/23/Annex-3 for reducing accumulation of sediments.

1. General principles
   - horizontal surfaces to be avoided wherever possible;
   - where longitudinals are fitted with face bar stiffeners, consideration should be given to fit the face bar stiffeners below the horizontal surfaces to aid drain off from the stiffeners;
   - arrange for induced flows of water, either by pump forces or gravitational forces, to wash along horizontal or near horizontal surfaces so that it re-suspends already settled sediment;
   - where horizontal stringers or webs are required, drainage holes to be as large as possible, especially if edge toe-stops are fitted where horizontal stringers are used as walkways, to encourage rapid flow of water off them as the water level in the tank falls;
   - internal girders, longitudinal, stiffeners, intercostals and floors, where fitted, should incorporate extra drain holes which allow water to flow with minimal restriction during discharge and stripping operations;
   - where inner members butt against bulkheads, their installation should be such as to prevent the formation of stagnant pools or sediment traps;
   - Scallops should be located at the joints of the inner bottom (tank top) longitudinal or intercostals and floors to allow for good airflow, and thus drying out of an empty tank. This will also allow air to escape to the air pipe during filling so that minimum air is trapped within the tank;
   - pipeline systems should be designed such that, when deballasting, disturbance of the water in the tank is as powerful as possible, so that the turbulence re-suspends sediment; and

2. Any designs depending upon water flow to re-suspend sediment should, as far as possible, be independent of human intervention, in order that the workload of ships' crews is minimal when operating the system.

3. The benefits of design concepts for reducing sediment accumulation are that there is likely to be good sediment removal while deballasting, with minimum retention of sediment in the tanks, and therefore a reduction or no need for removal by other means.

4. The design of all ships should provide safe access to allow for sediment removal and sampling.

CFD methodology can be implemented in following ways in relation with above mentioned points –

1. To simulate the flow of water inside ballast tanks. Suitable flow velocities corresponding to the pump flow can be accounted in such simulations
2. Simulate the effect of extra drain holes. The location of the drain holes can also be optimized with such simulations.
3. Assessment of pipelines with respect to the disturbance created during the deballasting.
4. Assessment of flow patterns in ballast water tanks for designing internal structure to provide effective flushing.

In the view of the above CFD tools are used for modelling sediments in ballast tanks. As this technology is relatively new to this field, the CFD model for sediment modeling is validated with the available data from experiments in the presented work. The details of validation studies are briefed in subsequent section.

**FLOW VISUALIZATION IN BALLAST TANKS**

In order to assess the sediments it is very much important to visualize the flow pattern in the tanks. The region where flow velocities are minimum or stagnant flows is called as dead zone. It is most likely that the sediments would be deposited in such zones. CFD methodology can be effectively used for finding such dead zones in the tank. Following example shows the flow pattern observed in the J-type tank.
Fig. 1 – Flow inside the J-tank: determining the dead zones

In Fig.1 inlet is at the bottom right corner and outlet is top left corner of the tank. Red colour indicating the highest flow velocity. As expected the dead zone in this case is bottom left corner of the tank. Flow in this region can be effectively improved by modifying internal framing/drain holes/lightening holes.

CFD MODELLING OF SEDIMENTS – VALIDATION STUDY

Problem definition
In order to validate the CFD model used for Sediment modeling slurry flow through a pipeline is considered. For this case a pipe of length 10m and a diameter of 0.495m are modeled. From inlet a flow mixture of sand and water are given with a flow velocity of 3.16 m/s. At the outlet the sediment deposition is measured and visualized along the length of pipe at observation planes (Fig. 2).

CFD Modeling
The CFD model used in this work is based on the extended two-fluid model, which uses granular kinetic theory to describe particle-particle interactions. Particles are considered to be smooth, spherical, inelastic, and to undergo binary collisions. The fundamental equations of mass, momentum, and energy conservation are then solved for each phase. Appropriate constitutive equations have to be specified in order to describe the physical and/or rheological properties of each phase and to close the conservation equations. The solids viscosity and pressure are computed as a function of granular temperature at any time and position. A more complete discussion of the extended two-fluid model, including the implementation of granular kinetic theory, can be found in Gidaspow. The Granular flow model used in the present study.

The volume-averaged, incompressible, transient Navier-Stokes equations are solved using STAR-CCM+ code. Mass and momentum equations were solved using a second-order implicit method for space and a first-order implicit method for time discretization. The conservation equations were discretized using the control volume technique. The discretization of the three-dimensional domain and the grid structure shown in Fig. 2.

Three dimensional transient simulations were performed. In these simulations, a constant time step of 0.001 s was used with time-averaged distributions of flow variables computed over a period of 10 s.

Results & Discussions

CFD simulations are carried out for two cases, both with pipe diameter of 0.495m, and flow velocity of 3.16 m/s. The sand particle size used in both cases is 165 µm. In the first case (Case-1) sand concentration is 0.104 and in the second case (Case-2) sand concentration is taken as 0.273. The sediment deposition variation along the length of the pipe at different observation planes for second case (Case-2) is given below.
In both the cases (Case 1 & Case2) the sand concentration at the outlet is measured along the depth of the pipe. A transient 3-D hydrodynamic model is developed for modelling sediments. A fair agreement is obtained between experimental data and the computed CFD results. The degree of asymmetry in the concentration profiles depends primarily upon the particle diameter, the mixture velocity, and the solids volume fraction. The CFD model described here is capable of predicting particle concentration profiles for fine particle slurries where fluid turbulence is effective at suspending the particles. It also performs satisfactorily when the particles are coarse and concentration profiles are primarily dependent upon the solids volume fraction.

It may be noted that the experiments to carry out the sedimentations study are expensive and uncertainties do exist in both CFD and experimental simulations. In general the sedimentation trend obtained in both cases is observed to be similar and CFD model used in present case can be considered valid for sedimentation study of ballast water tank.

**CFD MODELLING OF SEDIMENTS IN BALLAST TANK– CASE STUDY**

**Problem Definition**

A case study is performed in a ballast tank to improve the flow pattern inside the tank. Two cases were performed, without and with Drain holes. The flow patterns of sediments are studied in both the cases. The flow physics and the sediment modelling is used as of the above validation case. The tank geometry used in both the cases is shown as below.

**Results & Discussions**

CFD simulation is carried out for both the cases as above. A flow mixture of water and sediment is given with a flow velocity of 5m/sec at inlet. At outlet the flow mixture is allowed to exit. The solid volume fraction is plotted on the tank boundaries to study the effect of lightening and drain holes. The contour plots for both scenarios is given below.
From Fig. 8 it can be observed that for case without Drain holes most of the sediment is deposited in the tank near to inlet boundary and it decreases gradually as we go toward s outlet. Where as in the case of tank with Drain holes (Fig. 9) sediments deposition is spread uniformly and the rate of sediment flow at outlet for case with Drain holes is more to that of case without holes.

The present study can be extended to type approval of design of a ballast tank to provide credit for reduction of the sediment deposition in the tank. By reducing sediment within ballast tanks there are number of additional benefits aside from the reduction in invasive species. The key benefits include an increase in the deadweight allowance for cargo; this also manifests itself as a reduction in the overall unladen mass of the vessel which can result in increased fuel efficiency. Finally the corrosive effects of the sediment are minimized which increases the timeframe between applications of protective coatings and enables a more thorough structural inspection process to be conducted within ballast tanks.

APPLICATION OF CFD FOR BALLAST WATER TREATMENT PROCESSES

The vessels are required to be fitted with a Ballast Water Management System (BWMS) which has been approved in accordance with IMO guidelines for meeting the discharge performance standards (D-2) as specified in the BWM Convention.

The two most feasible treatment technologies for primary solids separation are filtration and cyclonic separation. Filter has operational problems regarding high-pressure drops and a strong tendency towards clogging and a low operational dependability. In contrast to filtration, spin particle separation is a relatively simple and inexpensive way of removing larger particles and organisms from ballast water.

Cyclonic separators are utilized in various industries which are relatively simple in construction and absence of moving parts mean that their capital and maintenance costs are lower than those of other control devices that are available.

CFD methodology can be used to simulate the effectiveness of various filtration techniques and is discussed below.

**CFD simulations of sediment removal with Cyclonic Separator**

The cyclone separator works by inducing spiral rotation in the primary phase (water and sediment mixture) imposing an enhanced radial acceleration on suspended particulate due to the centrifugal force. The density of the suspended particulate phase is normally greater than the primary phase. Due to the imposed swirll, larger particles migrate radially to the outer wall and then spiral down to the outlet.

In the following example Sediment-1 is lighter density particulate matter and Sediment-2 is relatively higher density particulate matter. Fig. 10 shows the deposited volume fraction of Sediment-1 and Fig. 11 shows volume fraction of Sediment-2. It can be noted that heavier sediments are deposited along the wall of the filter towards the bottom and the lighter sediment is spread throughout the filter. However the relatively fresh primary phase (water + Sediment-1) is discharged from the outlet at bottom. Thus the effectiveness of the cyclonic filter can be assessed with such CFD simulations.

**Heat treatment**

Most of these potential technologies haven’t yet been demonstrated in a full-scale shipboard environment. The effective filtration method can be achieved by using a combination of existing technologies together e.g. using the hydrocyclone as a primary treatment to remove larger organisms and/or suspended solids, and UV as a secondary treatment, to inactivate the remaining organisms, disinfect the ballast water and render it suitable to discharge.
The most viable option for secondary treatment at the present time is considered to be ultraviolet (UV) light irradiation. It has been the subject of laboratory testing on a range of marine organisms with positive results and it has already been used in other marine applications for many years.

The U-V treatment chamber consists of a circular duct intersecting a rectangular block containing an array of UV lights. The water in the duct flows through the stack and microorganisms are exposed to the high-frequency radiation and are destroyed. The geometric model of the system is shown in Fig. 12. At inlet water is allowed to flow with a flow rate of 70 kg/sec. The water and the microorganisms in the circular duct are then exposed to high frequency radiation from the U-V lamps which are placed orthogonal to the water flow. The U-V radiation from the lamps is set to be in the wavelength ranging from 2nm to 300nm. The rate of killing of microorganisms depends upon the selection of intensity of the U-V lamp.

In the present simulation micro-organism is modelled as particulate matter. The radiation intensity of the particulate matter before and after the exposure to the UV radiation can be seen in the Fig. 13. It can be observed that the radiation energy is increased in the direction of flow (after exposure to the UV radiation). This indicates the effectiveness of UV exposure. Higher the radiation energy lower is the micro-organisms density. Based on such studies various designs of UV chambers can be studied for an effective way for reducing the microorganisms.

CONCLUSIONS
From the above cases it can be inferred that CFD methodology can be used effectively for
1. Sedimentation studies in ballast water tanks, pipes.
2. Ballast water treatment process such as mechanical & UV treatments
The CFD capabilities for these applications are demonstrated with suitable examples. CFD (Computational Fluid Dynamics) model developed will provide the basis for the design of ballast tanks and detailed economic analysis of the filtration system for commercialization.

ACKNOWLEDGEMENTS
The authors express their earnest gratitude to the Indian Register of Shipping, Mumbai for supporting the reported work.
REFERENCES

Resolution MEPC 209(63), Annex-3, Guidelines on Design and Construction to facilitate Sediment control on Ships (G12), 2012


"ITTC - Recommended Procedures and Guidelines (2011)"
Practical Guidelines for Ship CFD Application. 7.5-03-02, pp 5