Studies on an Assessment of Safety with regard to the Damage Stability of Passenger Ships

Yoshitaka Ogawa and Kunihide Ohashi
National Maritime Research Institute, Japan

ABSTRACT

For the further clarification of safety with regard to the damage stability requirements on passenger ships, it is important to evaluate a safety level of damaged ships quantitatively. In this study, the method for evaluating the performance of cross-flooding arrangements of passenger ship is examined. A factor in terms of the pressure losses in the cross-flooding arrangements is evaluated by means of the Computational Fluid Dynamics (CFD). Through the comparison of present computation with experiments, the accuracy of the direct computation is clarified. In addition to this, a performance evaluated by the current IMO regulation is compared with the present computation.

KEYWORDS

Damage Stability; Cross-flooding arrangements; Passenger Ships; Computational Fluid Dynamics.

INTRODUCTION

For the further ensuring the safety in terms of damage stability, it is important to evaluate a safety level of damaged ships more quantitatively (e.g., Jalonen, 2010). Furthermore, it is considered that further validation of the evaluation method and tools is significant.

Based on this background, in this study, authors investigated the performance of cross-flooding arrangements. Firstly, a factor in terms of the pressure losses in the cross-flooding arrangements is evaluated by means of the Computational Fluid Dynamics (CFD). Secondly, model experiments for the validation of the computation are conducted. Through the comparison of present computation with experiments, the accuracy of the direct computation is clarified. Finally, a performance evaluated by means of the present computation and model experiments are compared with the current IMO (International Maritime Organization) requirement.

AN EVALUATION OF THE PRESSURE LOSS BY MEANS OF THE CFD

The purposes of the computation are firstly to validate the CFD tool with the experimental data and secondly to compare the accuracy of CFD computation with that of the simplified regression formulae in the IMO Recommendation (resolution MSC.245(83)) (IMO, 2007). In this study, the friction coefficients through the six units of structural ducts, which are divided by the girders with one or two manholes, are computed. The computed structures of cross-flooding ducts are shown in Figure 1. The sketch of girder in the structural ducts with one and two manholes is shown in Figure 2.

An outline of the CFD

The friction coefficients in cross-flooding ducts were computed by means of CFD tool developed at National Maritime Research Institute (NMRI), Japan. The model of the coordinate system of the CFD is shown in Figure 3. The cell number of the present CFD is shown in Table 1. The flow solver developed by NMRI (Hino, 2007), namely SURF (Solution algorithm for Unstructured RaNS with FVM), is used. The detailed features of SURF are follows;

- Treating unstructured polyhedral cells, hexahedra, tetrahedra, prisms and pyramids
- Coupling pressure and velocity by artificial compressibility approach
- Spatial discretization is based on a cell centered finite volume method

- Acceleration of convergence: Multi-grid method and local time stepping.

In terms of the turbulence model, following models are implemented :

- Spalart-Allmarass Model and its Modified
- k-ω/k-ε Blending Model (k-omega BSL,SST) and its DES-Org,F1,F2 Model.

The computational conditions are set as same as the model experiments. The velocity fields are non-dimensionalized by the inflow velocity at the entrance of the model setting. The length is non-dimensionalized by the duct height. The k-omega SST model is used as the turbulence model. The unsteady computation is carried out using dual time stepping consisted from the physical and pseudo time. The wall boundary is treated as low-Reynolds number model which means the minimum spacing on the wall is set as y+ nearly 1.0.

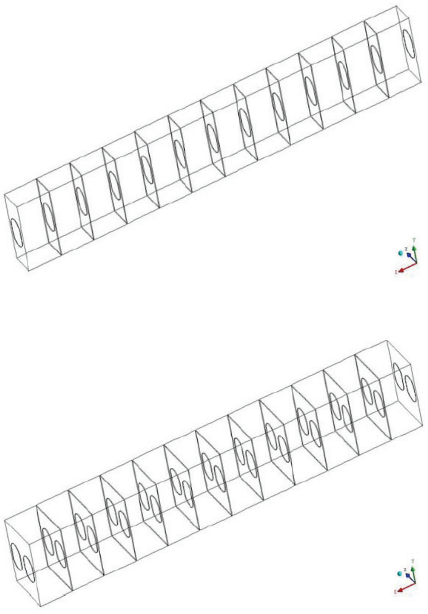


Fig. 1: A series of structural ducts with one and two manholes

Table 1: The number of cells

Number of units	Manhole/Position	Cells
6	1 hole/Center	987,040
6	2 hole	1,277,831

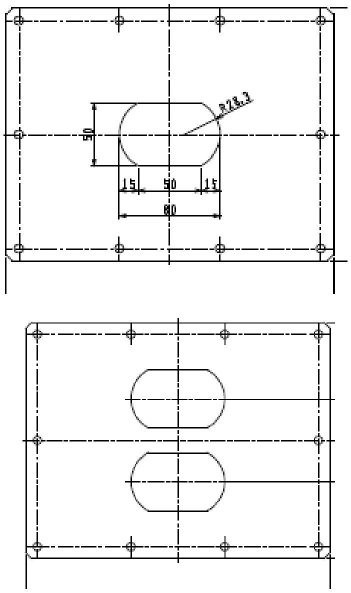


Fig. 2: A sketch of girder in the structural ducts

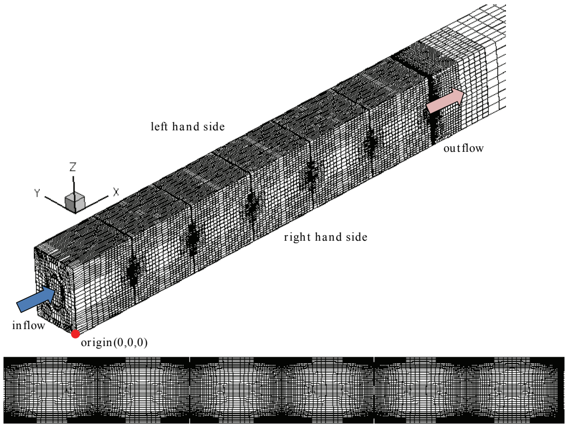


Fig. 3: The model and coordinate system of the CFD.

MODEL EXPERIMENTS

For the validation of computation, model tests were carried out with several variations of a cross-flooding duct design with one and two manholes. A setup of the six cross-duct modules is shown in Figure.4. The scale ratio to the subjected model in the CFD is 1:10. The flow was generated by means of the pump, which is also shown in Figure 4. A mass of inflow was kept constant in the experiments.

A photo of the six cross-duct modules is shown in Figure 5. Experiments were carried out in various conditions of flow in the ducts by adjusting the valve on the ducts. Total pressure and the friction coefficient were calculated based on the measured pressure and the measured stream velocity in the ducts. A pressure, *P*, was measured by the strain gauge. Stream velocity, *U*, was measured by the ultrasonic current meter. Total pressure, P_{TOTAL}, is evaluated by the combination of static and dynamic pressure, which is evaluated by means of the pressure and stream velocity. The friction coefficient at i-th unit, K_i, is evaluated by the variation of the total pressure at each duct.

$$P_{TOTAL} = P + 0.5\rho U^2 \qquad (1)$$

$$K_i = \frac{2(P_{TOTAL,i} - P_{TOTAL,i+1})}{\rho U_0{}^2} \qquad (2)$$

Here, ρ and U_0 describe the density of fluid and the stream velocity at the inflow respectively.

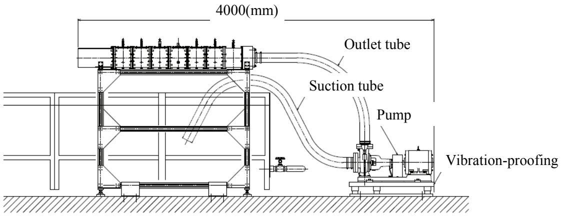


Fig. 4: A setup of the six cross-duct modules. (The 1:10 scale model).

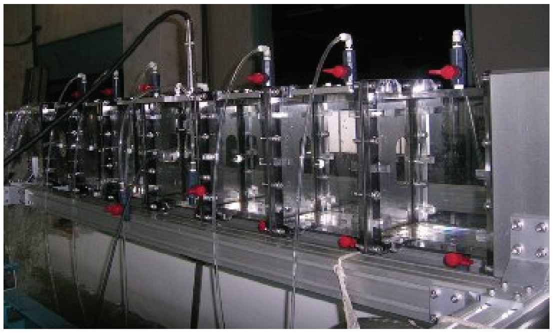


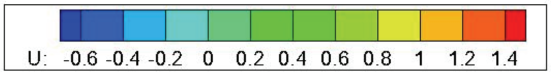
Fig. 5: A photo of the six cross-duct modules. (The 1:10 scale model).

COMPARISON OF COMPUTATION WITH EXPERIMENTS

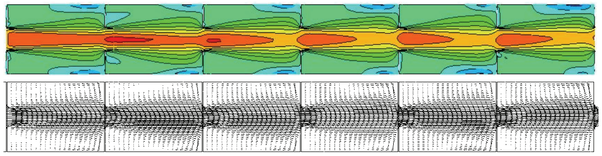
Overview of the CFD

Reynolds number is set as 4.0 x 10^5 and the physical time step size is 0.1. Figure 6 shows examples of computed axial velocity distribution and velocity vectors through the cross-flooding ducts. Velocity and pressure in figure 6 were time-averaged in 1000 computational steps which equal to 14 seconds with having dimension from the fully developed flow field.

It is found that water just passed through the manholes is accelerated. It is also found that water passed through one manhole is more accelerated than that through two manholes.



(a) 1 manhole



(b) 2 manholes

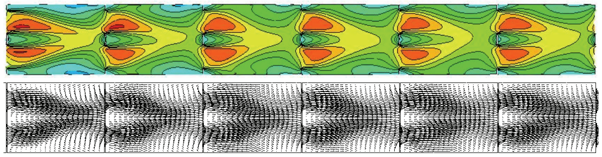


Fig. 6: Examples of the computed axial velocity distributions through the cross-flooding ducts

Comparison with Experiments and the Current IMO Requirements

The computed friction coefficients are compared with experiments. Results at each structural duct with one manhole and two manholes are shown in Figures 7 and 8. It is found that present computation well explain the experiments. It is clarified that friction coefficients becomes larger due to the difference of acceleration at manhole.

Furthermore, the computed friction coefficients are compared with the IMO requirements. The Maritime Safety Committee of the IMO, at its eighty-third session (MSC83), adopted the Recommendation on a standard method for evaluating cross-flooding arrangements (resolution MSC.245(83)). This recommendation, MSC.245(83), provides the method for assessing cross-flooding time through ducts and equalization arrangements, which presents various friction coefficients for water flow through pipes and valves. In addition, appendix 2 in this recommendation contains simplified regression formulae for pressure losses in cross-flooding ducts as a function of the distance between the girders.

On the other hand, in section 4 of the Recommendation, a direct calculation using computational fluid dynamics (CFD), time-domain simulations or model testing is allowed to be used as an alternative to the provisions in sections 2 and 3 of the recommendation. However, the comprehensive verification of applicability of a direct calculation has not necessarily been conducted. For the realization of a reliable direct calculation, it is important to verify the accuracy of the direct computations.

The friction coefficients computed by resolution MSC.245(83) is shown in Figures 7 and 8. It is also found that there are certain discrepancies between computation and the current recommendation, which implies that the recommendation requires different time from commencement of cross-flooding to the final equilibrium by means of the CFD computation and the experiments.

Although the regression formulae have been developed based on CFD computation (IMO, 2007), it is found that the difference of friction coefficients between the Recommendation and the CFD computation seems to be significant.

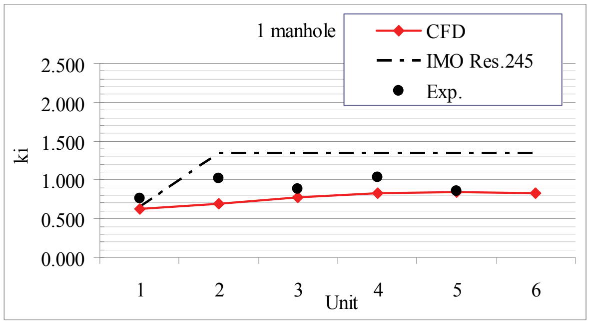


Fig. 7: The friction coefficient at each structural duct with 1 manhole

CLARIFICATION OF THE SCALE EFFECT

The computation in full scale is conducted to examine the scale effect for the pressure losses. Reynolds number is set as 4.0 x 10^6 and other conditions are similar with the model scale.

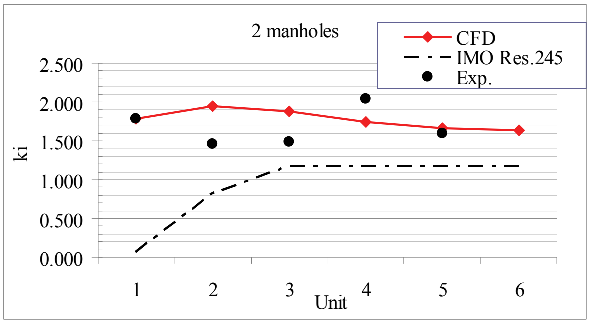
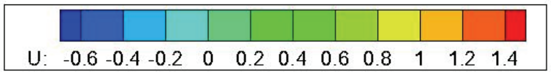


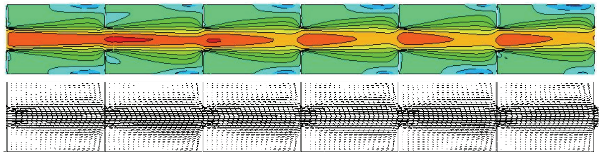
Fig. 8: The friction coefficient at each structural duct with 2 manholes

Figure 9 shows computed axial velocity distributions and velocity vectors of the model and full scale. Axial flows near the entrance of the cross flooding duct in the full scale are more accelerated than the flows of the model scale. The reversed flows can be observed in front of the girders. The regions of the reversed flows in the full scale become larger than the regions in the model scale, which yield in the acceleration of the axial flows of the full scale.

Figure 10 shows the comparisons of the friction coefficient. The coefficient of the full scale becomes abt.15% larger than the coefficient of the model scale. It is also found that the difference of friction coefficients between the Recommendation and the CFD computation seems to be significant.



(model)



(Full scale)

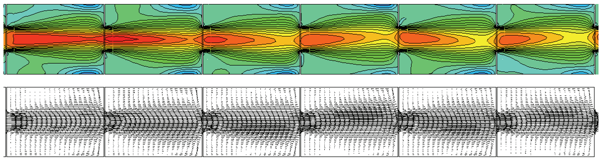


Fig. 9: The effect of Reynolds number on the axial velocity distributions through the cross-flooding ducts

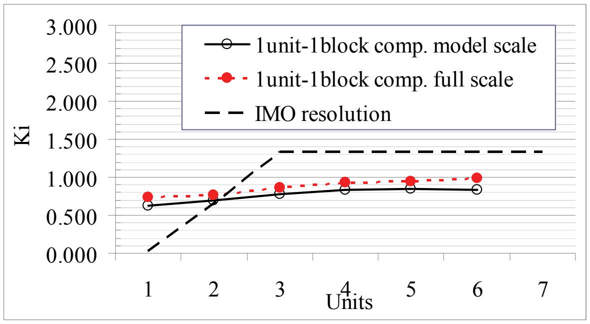


Fig. 10: The effect of scale on the friction coefficient at each structural duct with 1 manhole

CONCLUSIONS

1. The present computation explains the experiments well.
2. The flows inside of the cross flooding duct are visualized with the computational results. The reversed flows and flow acceleration near the girders are observed.
3. The friction coefficients of the present computation and experiment are different from that derived by means of the resolution MSC.245(83).
4. The scale effect for the friction coefficient is examined. The friction coefficient of the full scale becomes about 15% larger than that of the model scale.

For a quantitative conclusion, further study should be carried out based on the comprehensive computation under the various ducts. This will be done in the future study.

ACKNOWLEDMENTS

The present study was carried out in cooperation with the Japan Ship Technology Research Association through the Japanese project for the stability safety that is supported by the Nippon Foundation.

REFERENCES

Hino, T. (1997) A 3D Unstructured Grid Method for Incompressible Viscous Flows, J. of the Soc. Naval Archit. Japan, Vol.182

IMO MSC.245(83) (2007) Recommendation on a Standard Method for Evaluating Cross-Flooding Arrangements.

Jalonen, R., Jasionowski, A., Ruponen, P., Mery, N., Papanikolaou, A., Routi, A-L. (2010) FLOODSTAND – Integrated Flooding Control and Standard for Stability and Crises Management, Proc. of the 11th Intl Ship Stability Workshop, Wageningen, The Netherlands, 2010.