Time Domain Simulation of Cross-Flooding for Air Pipe Dimensioning

Pekka Ruponen

Napa Ltd & Helsinki University of Technology, Finland

Anna-Lea Routi

Aker Yards, Finland

ABSTRACT

An efficient method for the dimensioning of air pipes from the voids by using a novel time-domain flooding simulation tool is presented. The principles of the applied simulation method are briefly described and a thorough case study for the cross-flooding time in a U-shaped void of a modern large passenger ship design is reported. The presented method can be used for assessing sufficient dimensions for the air pipes. Furthermore, the required detail level for modelling of the cross-duct and the applied parameters for the pressure losses are discussed.

KEYWORDS

cross-flooding, simulation, air pipe, IMO Resolution A.266

INTRODUCTION

The IMO Resolution A.266 (VIII) for assessment of the cross-flooding times is currently being revised with special attention to the effect of counter air pressure in rooms, where the ventilation level is restricted. In IMO SLF50/WP.1 Annex 2 (2007), it is stated that full ventilation can be assumed if the area of the air pipes is at least 10 % of the area of the cross-flooding openings. Thus, for example the effect of the air pipe length is not considered at all for large pipes.

An efficient and practical method for assessing the cross-flooding time with all kinds of arrangements is dedicated time-domain flooding simulation, where the compression of air and the resulting airflows are properly taken into account. This kind of simulation is accepted as an alternative method for the assessment of cross-flooding time in the revised Resolution A.266.

Peters et al. (2003) have applied a time-domain flooding simulation and seakeeping tool for studying the effectiveness of various cross-flooding arrangements in naval ships. Vredeveldt and Journée (1991) and Xia et al. (1999) have considered cross-flooding and the counter air pressures with a simplified model. In this paper, a combination of these two approaches is presented, with relation to the revised Resolution A.266.

U-shaped voids are used in passenger ship designs since there are several benefits. Firstly, the voids give good protection against minor side damages, for example when steering the vessel into quay. Furthermore, they prevent unallowed oil leakage into sea as the voids form a kind of double hull. Due to these benefits, also the A-index is slightly increased. If the breadth of the side spaces is reasonably small, the U-shaped void does not significantly reduce the volume of the machinery space, and consequently, the design is feasible.

SIMULATION METHOD

Background

The presented cross-flooding calculations were performed with the NAPA Flooding Simulation tool, developed in close co-operation between Helsinki University of Technology (TKK) and Napa Ltd. The principles of the method are briefly described in the next section and in detail in Ruponen (2006). This implicit and iterative method allows time-accurate and efficient solutions for flooding problems that include airflows. The simulation method has been validated with dedicated model tests and some results have been presented in Ruponen et al. (2006).

Governing Equations

At each time step the conservation of mass must be satisfied in each flooded room, both for water and air. In general, the equation of continuity is:

$$\int_{\Omega} \frac{\partial \rho}{\partial t} d\Omega = -\int_{S} \rho \mathbf{v} \cdot d\mathbf{S} \tag{1}$$

where ρ is the density of the fluid, $\mathbf{v}$ is the velocity vector and $\mathbf{S}$ is the surface that bounds the control volume Ω. The normal vector of the surface points outwards from the control volume, hence the minus sign on the right hand side of the equation.

The velocities in the openings are calculated by applying Bernoulli's equation for a streamline from point A that is in the middle of a flooded room to point B in the opening. Consequently:

$$\int_{A}^{B} \frac{dp}{\rho} + \frac{1}{2}\left(u_B^2 - u_A^2\right) + g\left(h_B - h_A\right) = 0 \tag{2}$$

where p is air pressure, u is flow velocity and h is the height from the reference level. It is also assumed that the flow velocity is negligible in the center of the room ($u_A = 0$).

The equation (2) applies for inviscid and irrotational flow. The pressure losses in the openings and pipes are taken into account by applying semi-experimental discharge coefficients. Consequently, the mass flow through an opening is:

$$\dot{m} = \rho Q = \rho C_d A u \tag{3}$$

where Q is the volumetric flow through the opening, C_d is the discharge coefficient and A is the area of the opening. For pipes and ducts, the discharge coefficient is usually calculated from the sum of the pressure loss coefficients k_i, so that:

$$C_d = \frac{1}{\sqrt{1 + \sum_i k_i}} \tag{4}$$

The flooding process is assumed to be isothermal. Therefore, Boyle's law can be applied and the density of air is assumed to be linearly dependent on the pressure:

$$\rho_a = \frac{\rho_0}{p_0} p \tag{5}$$

where ρ_0 is the density of air at the atmospheric pressure p_0.

Pressure-Correction Technique

The ship model for flooding simulation can be considered as an unstructured and staggered grid (Fig. 1). Each modelled room is used as a single computational cell. However, the flux through a cell face is possible only if there is an opening that connects the rooms (cells). In this study, all openings were modelled as one-dimensional points with the given area and discharge coefficient. However, the same technique can easily be applied also for openings with more complex shape.

Water level in each room is considered to be flat and horizontal plane. Thus the sloshing effects are not taken into account.

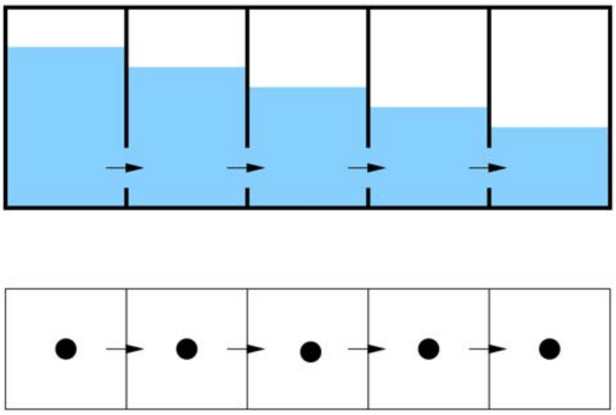

Fig. 1: Staggered grid in flooding simulation

Usually flooding simulation methods are based on the volumes of floodwater that are integrated explicitly from the flow velocities that are calculated from Bernoulli's equation. The water height differences are then calculated from the volumes of water with the heel and trim angles taken into account. However, the applied simulation method is based on a completely different approach, where the volumes are calculated on the basis of water heights and the heel and trim angles. This is reasonable as the water height is physically more meaningful than the volume of water since it represents the hydrostatic pressure. Consequently, the progress of the floodwater can be solved implicitly on the basis of the pressures in the rooms and the velocities in the openings.

The basic idea of a pressure-correction method is that the equation of continuity and the linearization of the momentum equation (Bernoulli) are used for the correction of the pressures until the iteration is converged and both the continuity and the conservation of momentum are satisfied at the same time.

Equation of Motion

This study concentrates on a statutory approach, where the damaged compartments are considered to be flooded immediately and only the cross-flooding to the equalizing side is calculated in time domain. Basically, this is just an improved and more accurate version of the formula in the Resolution A.266. The background for this simplified approach is presented in Solda (1961). Consequently, the motions of the ship are considered to be quasi-stationary and the sea is assumed to be calm.

CROSS-FLOODING EXAMPLE

Damage Case

The studied case is a U-shaped void, surrounding machinery spaces in a modern large passenger ship design. The tested damage case includes several compartments that are flooded immediately, and thus considered to be open to sea. Consequently, the initial heeling angle after the damage is large (10 degrees).

Cross-Duct

The sides of the U-shaped void are connected by a cross-duct that consists of four parts due to the longitudinal girders. In order to study the effect of the detail level in the modelling, three different models of the cross-duct were tested (Fig. 2):

- Level A: each girder defines a room partition
- Level B: cross-duct is considered as a single room
- Level C: cross-duct is considered as a single opening in the centerline

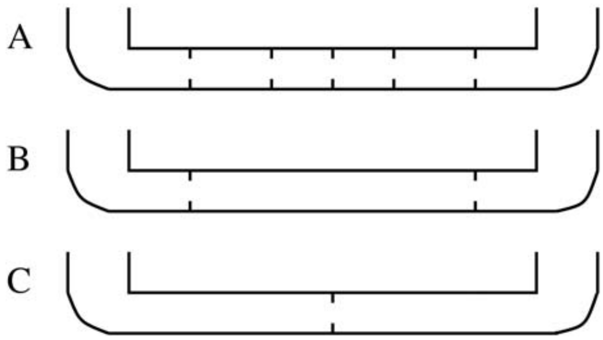

Fig. 2: Schematic representation of the different detail levels for modelling of the cross-duct

It is somewhat questionable what values should be used for the discharge coefficients of the openings. In general, $C_d \approx 0.6$ has usually been used in flooding simulations. However, this can be far too conservative when flooding through a cross-duct with several girders is considered. Most of the pressure losses are likely to take place in the inlet, where the discharging jet is initially formed. The subsequent openings will then have a smaller effect on the jet, and therefore, also the pressure losses are likely smaller.

In Level A, C_d = 0.65 is used for all openings in the duct. In Levels B and C, the applied discharge coefficients were obtained by using the formula in the proposed revision of the Resolution A.266 (IMO SLF50/WP.1 Annex 2, 2007). This is based on the RANSE computations, presented in Pittaluga and

Giannini (2006). In Level B, C_d = 0.65 was used for the opening on the damaged side and 0.45 for the other one and in Level C, C_d = 0.40 was used to represent the whole duct.

Air compression inside the cross-duct was ignored (Levels A and B) since the volume of the duct is minimal when compared to the volume of the whole U-void.

Air Pipes

Each side of the void is equipped with two separate air pipes. The pressure losses in these pipes were estimated by applying the formulae, presented in the Resolution A.266 (Table 1). Thus the effects of air compressibility were ignored. However, the effects of pipe diameter D and length L were taken into account. The unity for the outlet was excluded since it was already included in the calculation of the discharge coefficient, see equation (4). This procedure is in agreement with the proposed revision of the Resolution A.266 (IMO SLF50/WP.1 Annex 2, 2007).

Table 1: Applied pressure losses in the air pipes (according to Resolution A.266)

Component	**Value**
Friction	$k = 0.02\,L/D$
Inlet	$k = 0.51$
90 deg double mitre bend (2 pcs)	$k = 2 \cdot 0.43 = 0.86$

A constant time step of 0.2 s was used. The applied convergence criterion corresponds to a water height difference of 0.05 mm. It was checked that a shorter time step or a stricter convergence criterion did not have any notable effect on the results.

SIMULATIONS

Quasi-Stationary vs. Dynamic Heeling

In order to study the effects of the simplified approach with quasi-stationary motions, one case was simulated also with dynamic roll motion, using estimations for the added mass and linear damping. The comparison with quasi-stationary simulation is presented in Fig. 3.

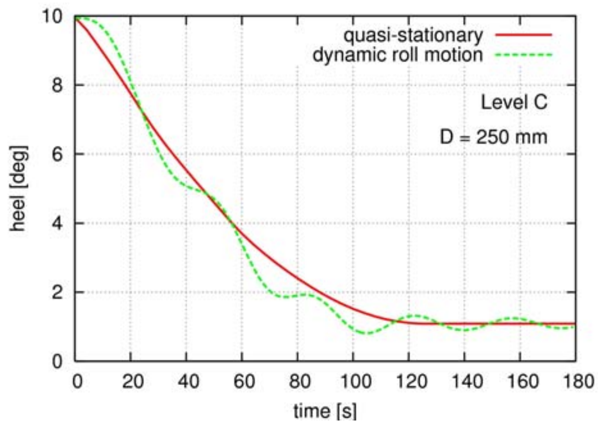

Fig. 3: Comparison of the heel angles with quasi-stationary method and dynamic roll motion

It seems that the quasi-stationary method results in slightly longer equalization times, and hence it is considered to be suitable for statutory calculations. In general, the results are very similar with both approaches.

Effect of the Cross-Duct Modelling

Comparisons of the heeling angle and over pressure in the equalizing side of the void are presented in Fig. 4 and Fig. 5, respectively. The equalization process is estimated very similarly with all tested modelling levels of the cross-duct. However, Level C gives much larger maximum over pressure in the void during the flooding. This may result from the assumption of full ventilation in the duct.

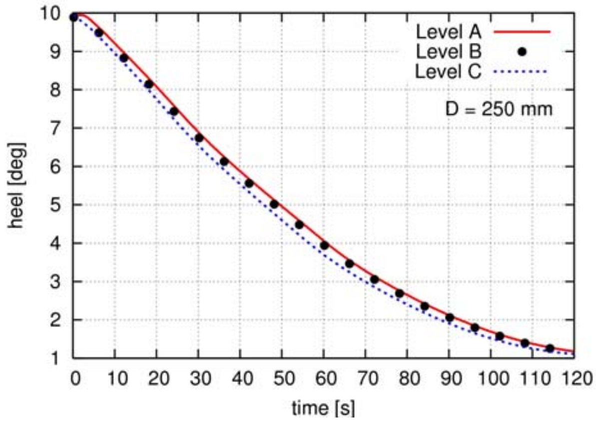

Fig. 4: Comparison of the time histories for the heeling angle with different modelling levels of the cross-duct

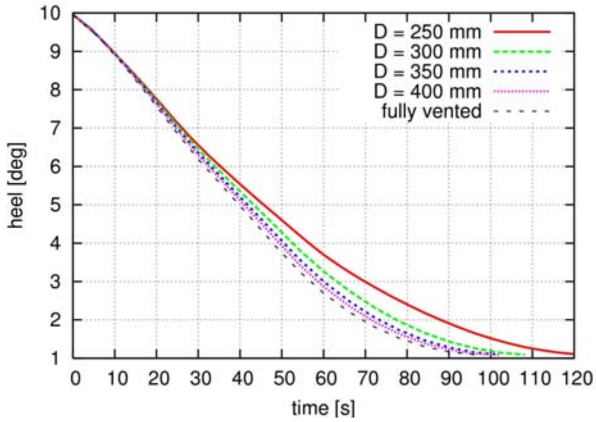

Fig. 6: Comparison of heeling angle with different sizes of air pipes

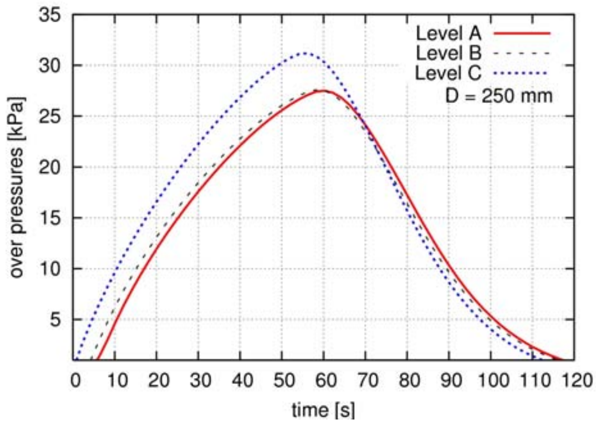

Fig. 5: Comparison of the time histories for over pressure in the equalizing side with different modelling levels of the cross-duct

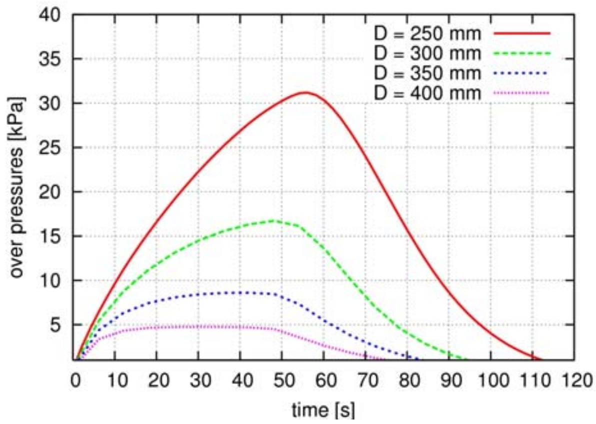

Fig. 7: Comparison of over pressure in the equalizing side with different sizes of air pipes

Air Pipe Dimensioning

The simulations for the dimensioning of the air pipes were performed with the cross-duct modelling Level C. The pipe diameter was increased until there was no difference, when compared to the results from the simulation with full ventilation in the void. The results for the heeling angle, over pressure and volume of floodwater in the equalizing side are presented in Fig. 6, Fig. 7 and Fig. 8, respectively.

The area of the largest pipe (D = 400 mm) is 9.9 % of the area of the cross-flooding openings. The results for the heeling angle and volume of water are very close to the results from the simulation with fully vented void.

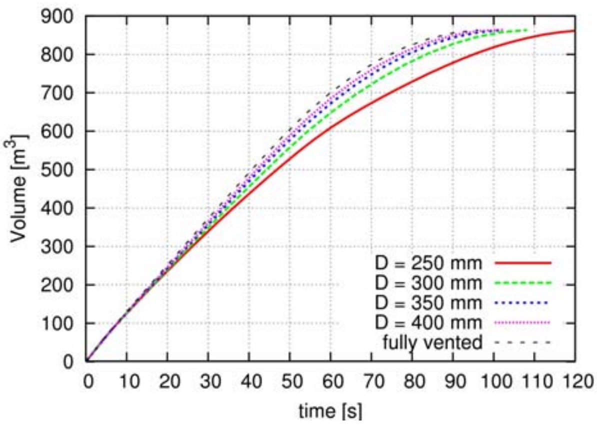

Fig. 8: Comparison of floodwater volume in the equalizing side with different sizes of air pipes

The effect of air pipe diameter is not notable in the beginning of the equalization process.

Thereafter, near the equilibrium condition, the effect is more significant.

On the basis of these results, it seems that in this particular case, the criterion of 10 % area for the air pipes in the revised Resolution A.266 seems to be reasonable. However, no general conclusions can be made without systematic simulations with many different ship designs.

Realistic Damage Case

The presented simulations were based on the statutory approach, where the damaged rooms are considered to be immediately flooded. For comparison, a separate case, where only the U-void is damaged, was calculated both with the statutory approach and with a modelled damage opening of 5.0 m^2. Simulations were performed with two different air pipe diameters.

The results for the time histories of heeling are presented in Fig. 9. The formula of the Resolution A.266 gives an equalization time of 131 s, which is slightly conservative, when compared to the simulations.

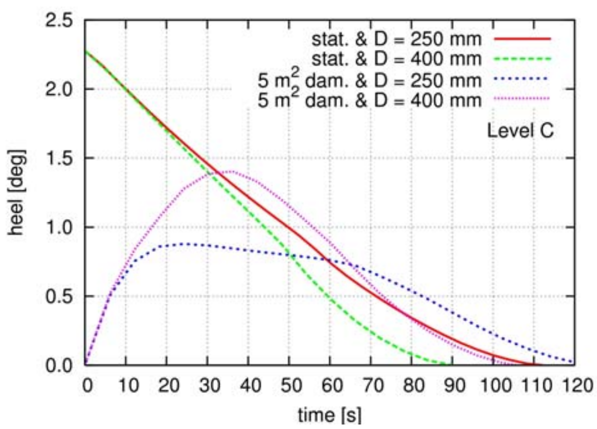

Fig. 9: Comparison of heeling angle with the statutory case and with 5.0 m^2 damage opening, simulated with two different air pipe diameters

When also the damage opening is modelled, the compression of air in the damage side of the void slows down the flooding from the sea. Consequently, the maximum heeling angle is decreased with smaller air pipes, even though the equalization time is longer. It might be reasonable to take this phenomenon into account with damages to rooms that are located far below the water line.

CONCLUSIONS

Time-domain flooding simulation, with the compression of air taken into account, seems to be a very efficient and practical tool for assessing the cross-flooding times more realistically than the simplified formula in the Resolution A.266.

However, it is obvious that more dedicated CFD computations and large scale model tests are needed in order to develop and verify the simple formulae for pressure losses in various cross-flooding arrangements. Furthermore, experimental and computational data on the pressure losses for airflow in the ventilation pipes are needed. In the presented simulations it was assumed that the formulae for water flow in pipes could also be used for assessing pressure losses in the case of airflow. However, dedicated studies are needed in order to justify this procedure.

Fully quasi-stationary calculation of the ship motions seems to be justified in the statutory approach since there is no transient heeling in the start of the flooding, unless the cross-duct is very large and the cross-flooding time is minimal. On the other hand, in such a situation, accurate assessment of the cross-flooding time is not needed.

In realistic flooding simulation, the heeling angle approaches the equilibrium condition almost asymptotically. The last seconds of the equalization are definitely insignificant, and therefore, some generally accepted criterion for the practical limits of the equilibrium should be established.

REFERENCES

IMO SLF50/WP.1 Annex 2 2007. Recommendation on a Standard Method for Evaluating Cross-Flooding Arrangements.

Peters, A. J., Galloway, M., Minnick, P. V. 2003. Cross-Flooding Design Using Simulations, Proceedings of the 8th International Conference on Stability of Ships and Ocean Vehicles, Madrid, Spain, 2003, pp. 743-755.

Pittaluga, C., Giannini, M. 2006. Pressure Losses Estimation for Structural Double Bottom by CFD Technique, CETENA Technical Report, published in internet (cited 14.5.2007), http://www.sname.org/committees/tech_ops/O44/sdsiscg/49/A266-1.pdf and http://www.sname.org/committees/tech_ops/O44/sdsiscg/49/A266-2.pdf

Ruponen, P. 2006. Pressure-Correction Method for Simulation of Progressive Flooding and Internal Air Flows, Schiffstechnik – Ship Technology Research, Vol. 53, No. 2, pp. 63-73.

Ruponen, P., Sundell, T., Larmela, M. 2006. Validation of a Simulation Method for Progressive Flooding, Proceedings of the 9th International Conference on Stability of Ships and Ocean Vehicles, Rio de Janeiro, Brazil, 25-29.9.2006, Vol. 2, pp. 607-616.

Solda, G. S. 1961. Equalisation of Unsymmetrical Flooding, Transactions of Royal Institute of Naval Architects, RINA, Vol. 103, pp. 219-225.

Vredeveldt, A. W., Journée, J. M. J. 1991. Roll Motion of Ships due to Sudden Water Ingress, Calculations and Experiments, RINA'91, International Conference on Ro-Ro Safety and Vulnerability the Way Ahead, London, United Kingdom, 17-19. April 1991, Vol. I.

Xia, J., Jensen, J. J., Pedersen, P. T. 1999. A Dynamic Model for Roll Motion of Ships Due to Flooding, Schiffstechnik – Ship Technology Research, Vol. 46, pp. 208-216.