Investigation of dynamic characteristics of the flooding water of the damaged compartment of an ITTC RORO Passenger

Cho, Seok Kyu*, Hong, Sa Young*, Kim, Yoon Ho**, Lee, Kyu Jung**

*Maritime and Ocean Engineering Research Institute, KORDI, Korea

**Department of Mechanical Engineering, Korea University, Korea

Abstract

When a ship is damaged and flooded, the motion of the damaged ship is significantly influenced by the flooding water dynamics. The flooding water in the damaged ship has been treated as a lumped mass under the quasi-static assumption in most of previous researches. To calculate the motion of damaged ship rigorously, it is necessary to analyze the coupled dynamics of flooding water. In this study, a series of numerical and experimental studies is conducted for the damaged part of ITTC RORO passenger. FLOW3D is used for investigating the feasibility of the state of the art CFD technique. An applicability of the coupled motion analysis of damaged ships can be confirmed by agreement between the numerical results and the model experiments. A CFD technique is considered as a numerical modeling of the dynamics of flooding water.

1. Introduction

The stability of a damaged ship is an important design factor for RORO passengers, large container ships and naval vessels. The parameters of damaged ship motion are mass inertia, added mass, restoring force, potential and viscous damping, wave induced forces, flooding process and coupled ship motion due to flooding water. Among these parameters, flooding water dynamics is the most important and difficult one to be analyzed because of the complexity of free surface motion in the coupled ship motion. Previous researches(Palazzi 2004, Letizia 2004 and MA 2000) on the flooding water dynamics used the hydraulic model that regards the flooding water as a lumped mass. Such a quasi-static modelling has the advantages of the flexibility of modeling and the simplicity of implementation for various conditions(Papanikolaou 2002). If a damaged compartment is complex and flooding water is coupled with ship motion, however, this method is not capable of predicting the coupled motion dynamics accurately. To assess the effect of various damaged inlets and internal structures on flooding water dynamics, it is necessary to use CFD technique(Woodburn 2002, Papanikolaou 2002). The CFD requires very high computing power and long simulation time. Advances of computer technology and numerical method for free surface analysis have made it possible to use the CFD method simulating the flooding phenomena.

In this study, an experiment and numerical simulation for the damaged part of RORO passenger is conducted to investigate accuracy of the CFD results on flooding water dynamics quantitatively. 6-dof forces and moments by flooding water are measured for various combinations of internal structures and damaged inlets such as geometry, position and size. The FLOW3D is used for numerical simulations and the numerical results agree with the experimental results.

2. Model Test

The PRR02's damaged compartment is adopted as the present model which already used in the 24th ITTC Benchmark Study on numerical prediction of damage ship stability in waves. The model was made of transparent acrylic plate to observe the incoming floodwater and the geometry of model is shown in Fig. 1 and the main dimensions are summarized in Table 1. To investigate the effect of inner compartments, we made two models as shown in Fig. 2. The one is a simple model with the inside empty; the other is a real model having internal compartments and engine blocks. The effect of damaged inlets is also investigated by changing the geometry, size and position of the inlet as shown in Fig. 3. Measurement items are forces, moments and high speed captured flow images. The summary of experiments is below.

▷ Models : Simple model and Real model(Fig. 2)
▷ Damaged inlets : Inlet1, Inlet2, Inlet3, Inlet4, Inlet5, Inlet6(Fig. 3)
▷ Load measurement : Force and moment(ATI 6 Axis Loadcell), 50 Hz
▷ Flow measurement : High speed camera(Redlake), 250 frame/sec

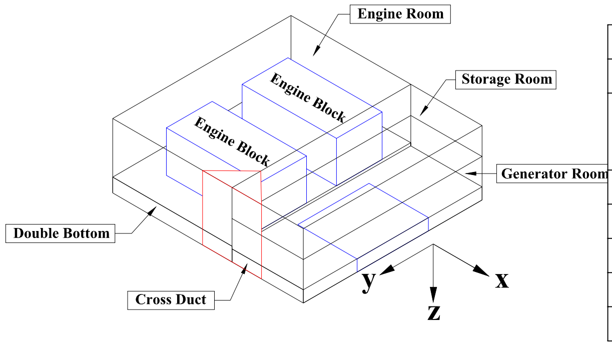


Fig. 1 Geometry of damaged part

Table 1

Model scale	48.57
Damage side	Starboard
Damaged length (SOLAS guide)	8.244 m (0.03×L+3.0)
Length	26.714 m
Breadth	25.000 m
Height	9.025 m
Draft	6.400 m
Engine Blocks	13.18×6.4×5.73


Fig. 2 Model(Left : Simple model, Right : Real model)


Fig. 3 Inlet Geometry(Inlet1, Inlet2, Inlet3, Inlet4, Inlet5, Inlet6)

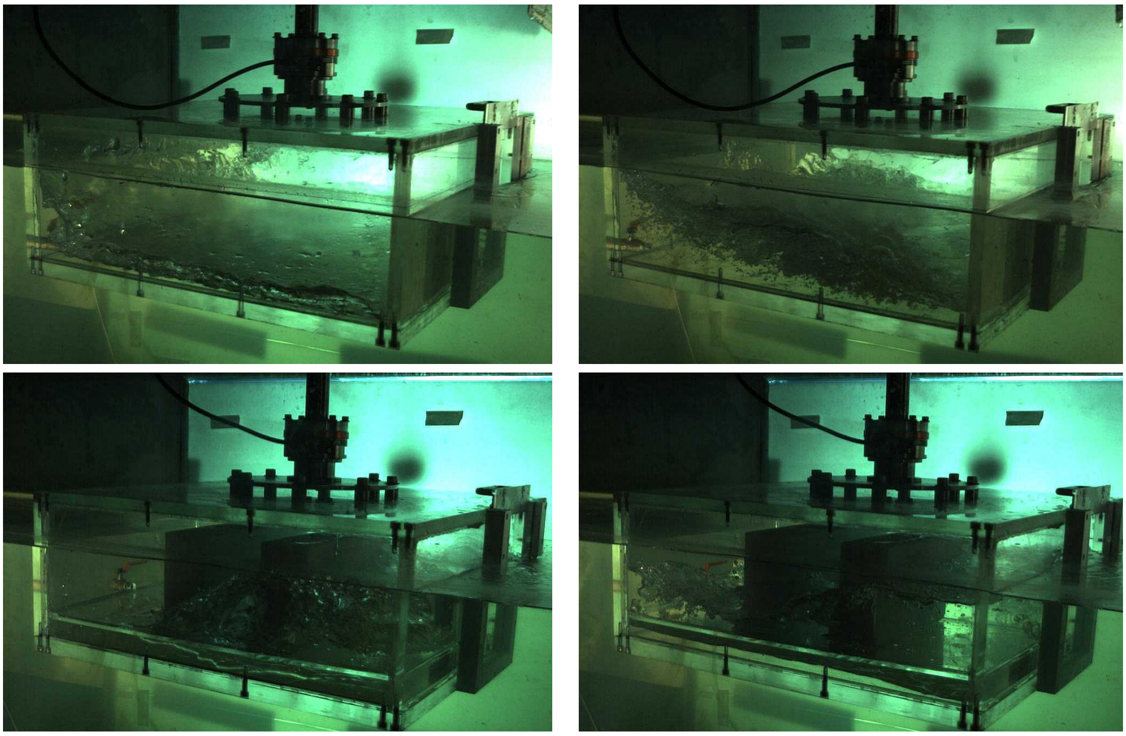


Fig. 4 Photos of model test(Above: Simple model - Inlet 1, Below: Real model - Inlet 1)

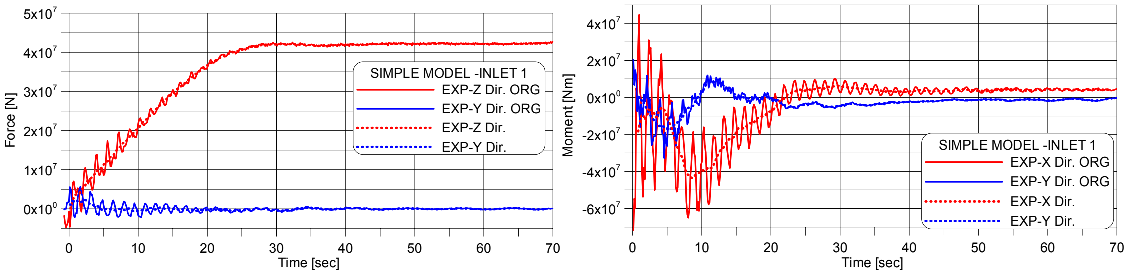


Fig. 5 Force and moment of Simple model - Inlet 1

The instantaneous images of high speed camera are displayed in Fig. 4 and the forces and moments of the simple model - Inlet 1 are demonstrated in Fig. 5. The damaged inlet is opened by pulling the slide and the time of opening is about 0.12 seconds. When the slide is pulled, the model is oscillated with natural frequency of the measuring system as shown in Fig. 5. To exclude the unwanted affects we have taken moving average. The solid lines are original oscillating signals and dashed lines are averaged lines. The results of model tests are compared with FLOW3D results in chapter 4.

3. Numerical Simulation

3.1 Numerical scheme of FLOW3D

The FLOW3D is a commercial code calculating general fluid problem and is based on FDM(Finite Difference Method), FVM(Finite Volume Method) and VOF(Volume Of Fluid) method. The governing equations used in FLOW3D are as below.

The continuity equation

$$\frac{\partial \rho}{\partial t}+\nabla \bullet (\rho \mathrm{U})=0$$

The momentum equation

$$\frac{\partial U}{\partial t}+U\bullet\nabla U=-\frac{1}{\rho}\nabla P+G+\nu\nabla^2 U+F$$

where ρ is density, ν is viscosity, G is gravity force and F is external body forces.

VOF advection equation solving the free surface is next.

$$\frac{\partial F}{\partial t}+\left\{\frac{\partial}{\partial x}(Fu)+\frac{\partial}{\partial y}(Fv)+\frac{\partial}{\partial z}(Fw)\right\}=\mathrm{FDIF}+\mathrm{FSOR}$$

where F, FDIF and FSOR denote VOF function, diffusion of fluid fraction and fluid source/sink. F is defined in Fig. 6.

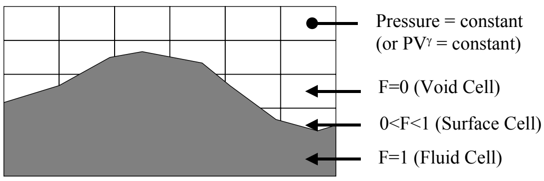


Fig. 6 Definition of VOF function(F)

3.2 Modeling of RORO Passenger's damaged part

We construct the meshes and boundary conditions same as the model test conditions for FLOW3D analysis. The computation time is about the 144~200 hour using Intel Pentium 4 2.8 GHz CPU.

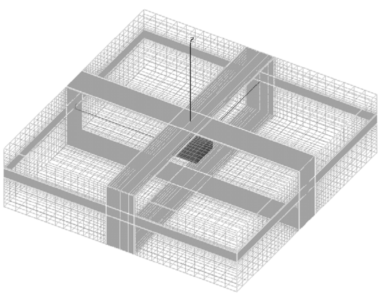


Fig. 7 Grid system

Table 2 Grid

Grid	3D/Structured grid		
Mesh	X	120~130	Total : 900,000 ~1,300,000
	Y	120~130	
	Z	60~90	
Computation time : 144~200 hour / P4 2.8 GHz			

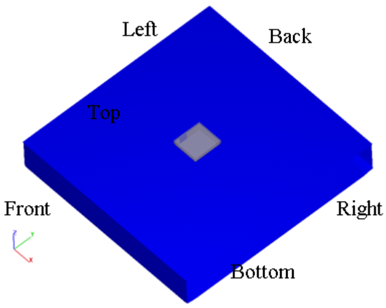


Fig. 8 Boundary set-up

Table 3 Boundary conditions

Top	Pressure	1 atm
Bottom	Wall	
Left/Right	Outflow	Water level : 6.4 m
Front/Back	Outflow	Water level : 6.4 m

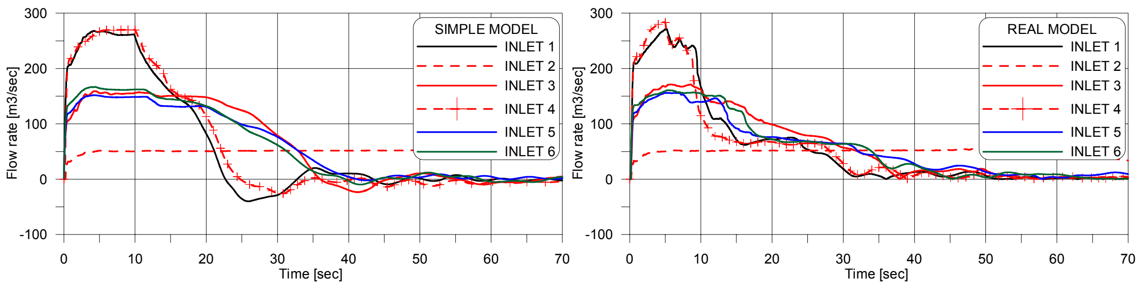
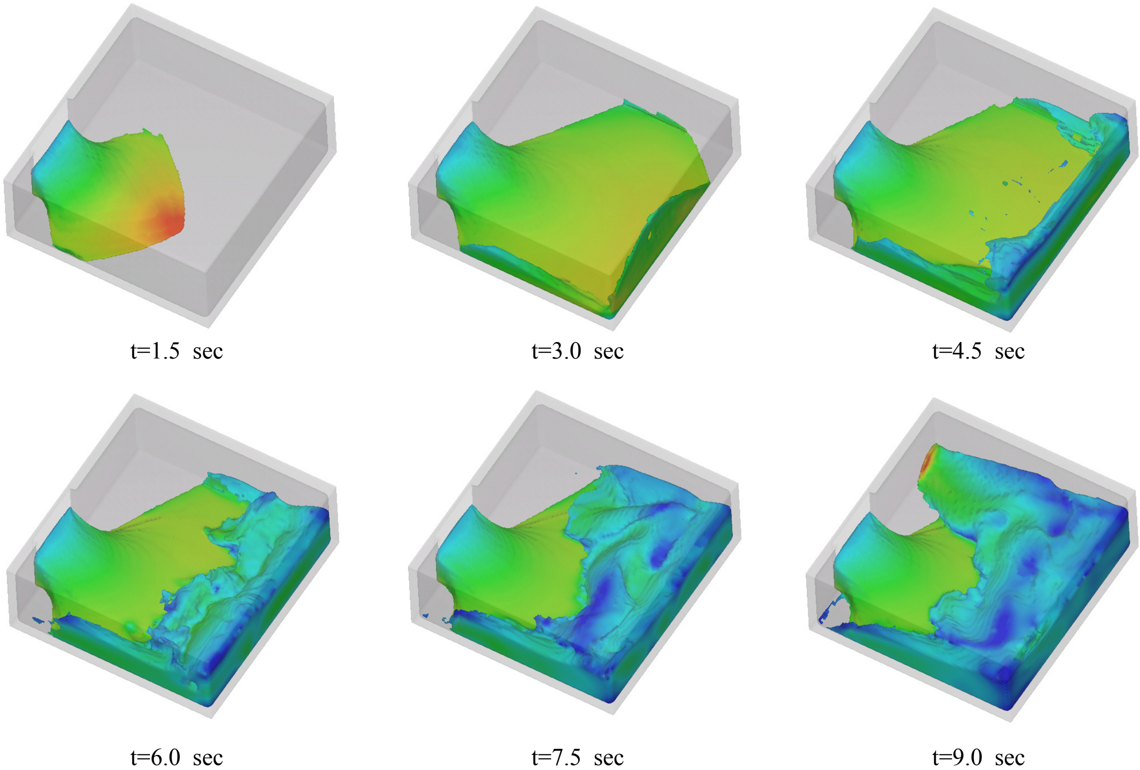


Fig. 9 Flow rate results(Left : Simple model, Right : Real model)

Fig. 9 represents the calculation results of flow rates. Ingresses terminate at the time 40 seconds after the inlet open except Inlet2 condition. It is observed that backflow is generated for the simple model. From the comparison the results of Inlet1, Inlet4 and those of Inlet2, Inlet3, Inlet5, flow rates is determined by the size of inlet under the free surface. The effect of geometry of inlet and inner compartments is small. Fig. 10 is the snapshot of the Simple model - Inlet 1 and the Real model - Inlet 1. For the Simple model - Inlet 1 the flooding water reaches to the opposite side of the inlet and accumulates after 3 seconds, and the flow is back to the inlet side generating the rotational flow at time of 10 seconds. For the Real model - Inlet 1 the flow characteristics is similar to the Simple model in the Generator room and the Storage room. But the flooding water in the Engine room accumulates in the inlet side and flows to the inner side due to the Engine blocks.

☐ Simple model - Inlet 1



☐ Real model - Inlet 1

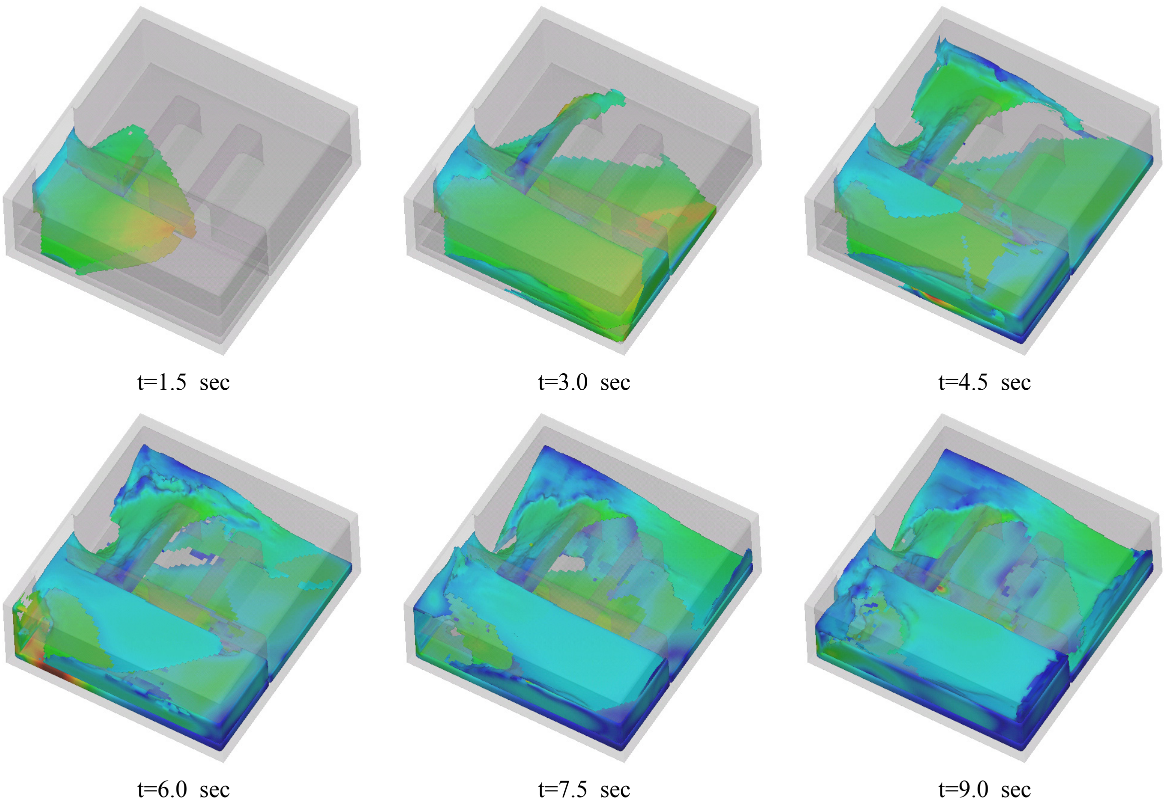


Fig. 10 Snapshot of Flow3D analysis

4. Results(comparison of experiment and numerical simulation)

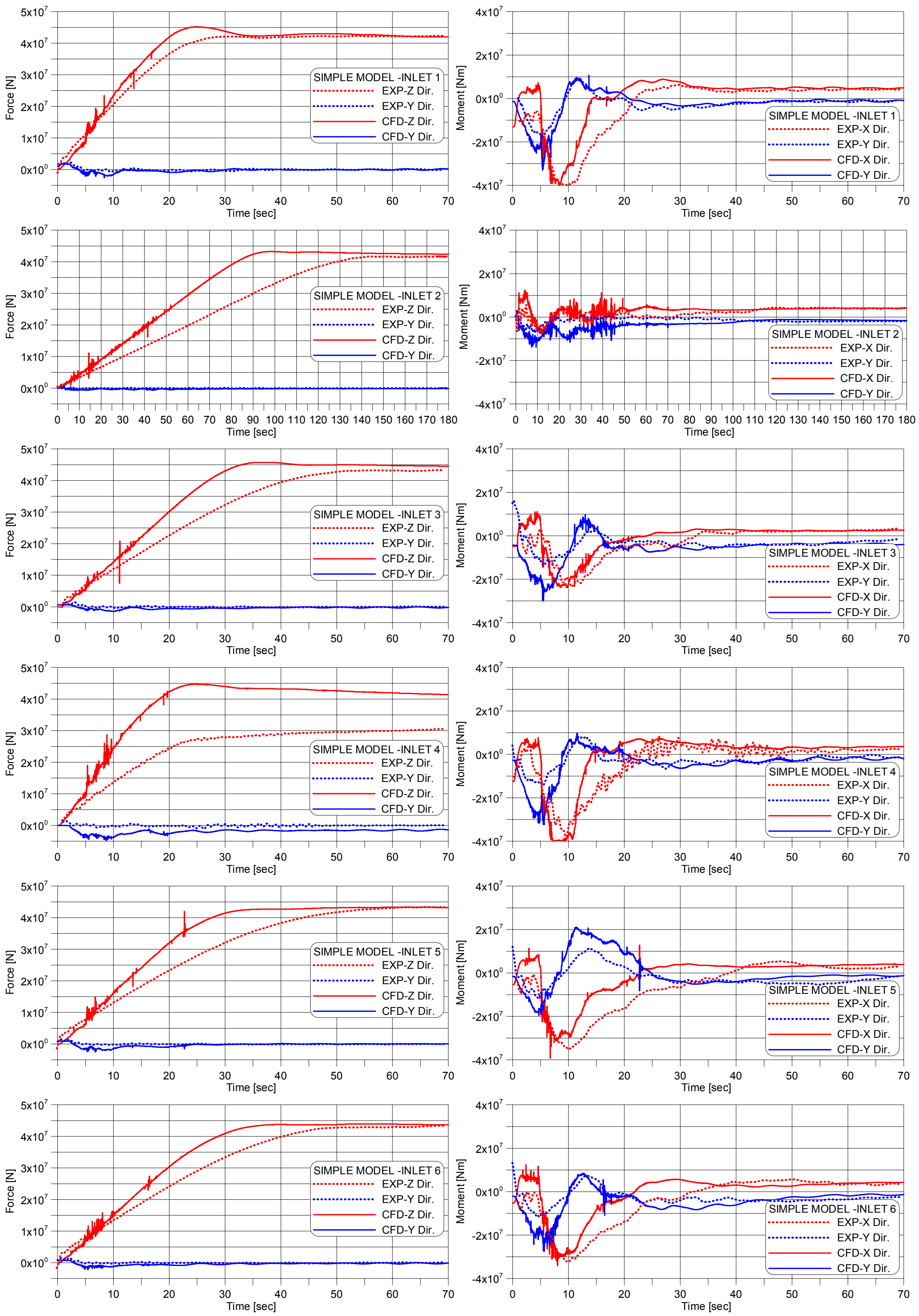


Fig. 11 Comparisons of experiment and numerical simulation for Simple model

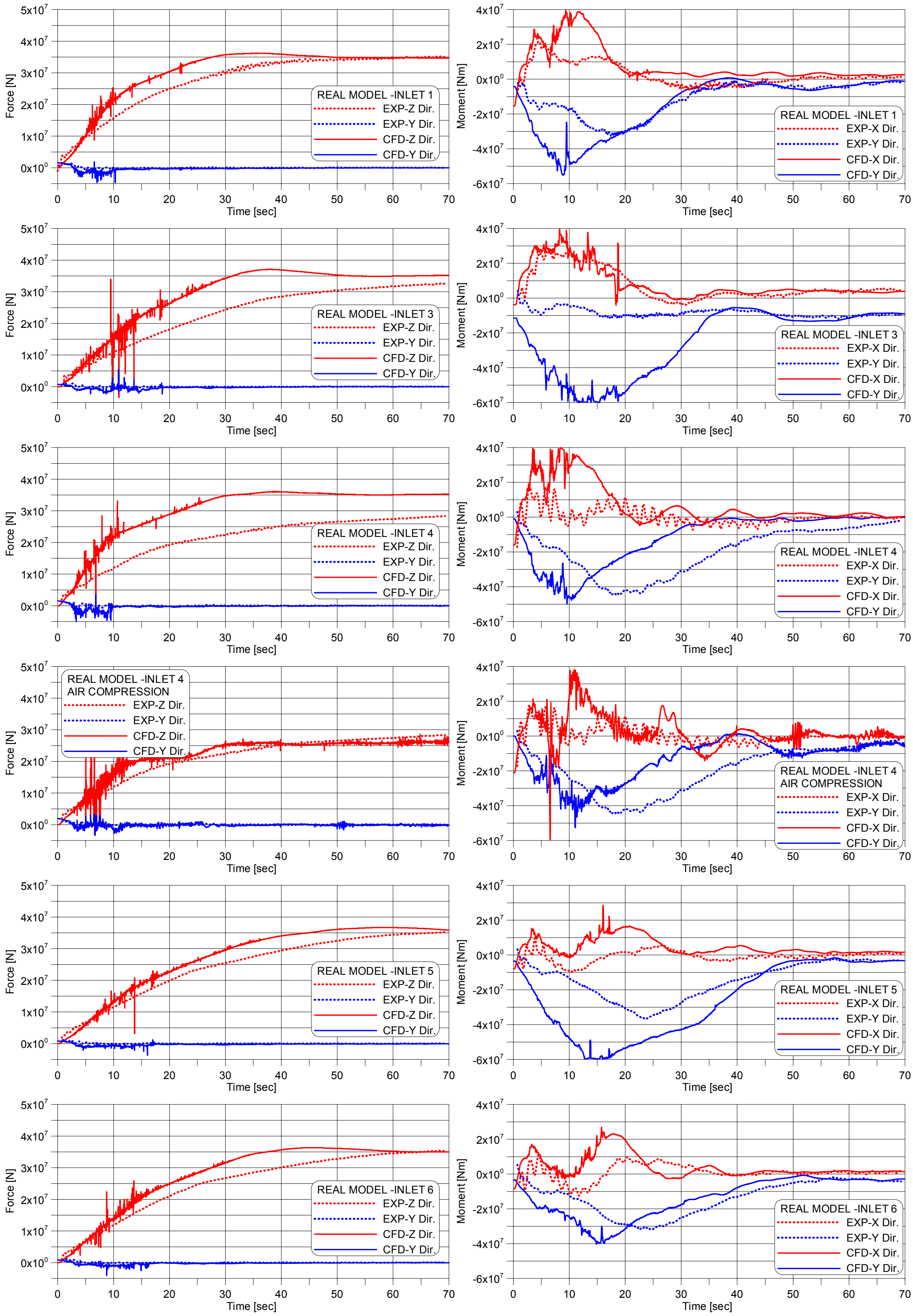


Fig. 12 Comparisons of experiment and numerical simulation for Real model

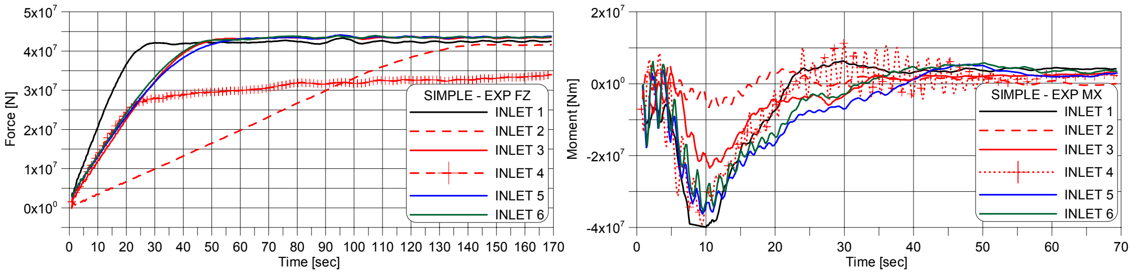


Fig. 13 Experimental Results of Simple model

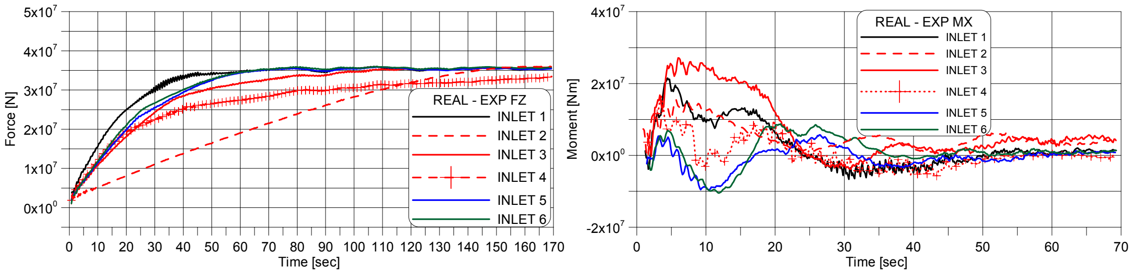


Fig. 14 Experimental Results of Real model

The Results of the experiments and the numerical simulations are shown in Fig. 11 to Fig. 14.

▷ Simple model results(Fig. 11, Fig. 13)

○ *Force*

The Z direction force represents the ingress magnitude indirectly and the ingress of flooding water in FLOW3D is faster than the experiments. We consider this is because of the air compression and viscous effect not treated well in the FLOW3D. In case of Inlet4 the results of experiment and FLOW3D make a big difference. Inlet1 is below free surface and flooding water compresses the air trapped inside the chamber. The compressed air makes the ingress magnitude smaller than other inlet conditions. The FLOW3D can simulate the air compression in this case. When the inlet is open to air, we can ignore the compressibility of air. But we must consider the effect of air when the inner space is confined.

○ *Moment*

The moments have a same tendency but a difference in quantity. The X direction and Y direction moment stands for roll and pitch moment. Roll moment have a positive value initially and become a negative value and finally reach equilibrium. This characteristics agree well with flow dynamics(flooding process : accumulation at inlet side of inlet → accumulation at opposite side of inlet). Pitch moment is generated by circulating flow. We see the moments of Inlet1, Inlet4, Inlet5 and Inlet6 are similar and Inlet3 is intermediate and Inlet2 is the smallest in Fig. 13. From this we are aware that the roll moment is influenced by the area of inlet under free surface and the compressibility of air.

▷ Real model results(Fig. 12, Fig. 14)

○ *Force*

The result of Inlet4 considering the air compression agree well with experiment. The compressibility of air is treated by Gas law, $PV^{\gamma} = C$ where γ is 1.4. The tendency of Z force is similar to Simple model.

○ *Moment*

The tendency of roll moment is different with the Simple model. Roll moment have a positive value and reach equilibrium. This feature is explained by the fact the flooding water accumulates in inlet side and flows to inner side due to the Engine blocks. Pitch moment have a negative value because ingress magnitude is small in the engine room due to engine blocks while it is large in the storage room and the generator room. In case of Inlet3 the discrepancy of pitch moment may be caused by the different gap size effect of inlet and inner structure. From the Fig. 14 we can verify roll moment is decided by not only the area of inlet under free surface but also the inlet geometry. If damaged part has internal compartments and structures, roll moment is influenced by the combination of the area of inlet under free surface and the inlet geometry

5. Conclusions

From the experiment and numerical simulation of RORO passenger's damaged compartment, we confirmed the next results

● Comparing the experiment and FLOE3D simulation, the feasibility of CFD technique to calculate the flooding water in a damaged ship is verified.

● Flow rate : From the results of FLOW3D, flow rates is determined by the area of inlet under free surface.

● Force : FLOW3D predicts well the experiments and it is necessary to consider the compressibility of air and viscosity in FLOW3D for accurate results

● Moment : The tendency of FLOW3D is similar to the experiments. Moment is influenced by the flooding water distribution and the characteristics of flooding water is decided by the area of inlet under free surface, the internal structures of damaged part and the inlet conditions.

Acknowledgement

This study is a part of the research program, "Development of Simulation technologies for Dynamic Stability of Ships" supported by Ministry of Commerce, Industry and Energy of Korea.

References

Palazzi L. and De Kat J. , 2004, "Model Experiments and Simulations of a Damaged Ship With Air Flow Taken Into Account", Marine Technology, Vol. 41, No. 1, pp. 38-44

Letizia, L., Vassalos, D. and Jasionowski, A., 2004, "“New Insights into Ship-Floodwater-Sea Dynamics", International Shipbuilding Progress, Vol 51, No 2/3, pp. 273~291

MA Y. , Katayama T. and Ikeda Y., 2000, "A Study on Stability of Damaged in Intermediate Stage of Flooding", J. Kansai Soc. N. A., No. 234, pp. 179~186

Woodburn, P., Gallagher, P. and Letizia, L., 2002, "Fundamentals of Damaged Ship Survivability", RINA Transactions, pp. 143~163

Papanikolaou, A. and Spanos, D., 2002, "On the Modelling of Floodwater Dynamics and Its Effects on Ship Motion", Proc. 6th International Ship Stability Workshop, Glen Cove