論文の内容の要旨
Abstract of Dissertation

論文題目
Title of Dissertation

Analysis of the Mechanism of Slamming on the Bow Flare Region of Ship's Hull by Using RaNS CFD Method

(RaNS 流体数値計算による船首フレアスラミング機構の解析)

氏　　名　　　ラーマン モハメド マシユール
Name of Author　　Md. Mashiur RAHAMAN

（本文）**(Abstract)**

Modern container ship has large flare on the bow flare region for the purpose of accommodating more containers and their safe handling. This flare region on the forward part of the ship's hull surface experiences hydrodynamic load in actual operating conditions in waves. Traditionally, strength requirements in the bow flare region is evaluated based on the empirical formula proposed by various classification societies. Study reveals that a sizeable difference occurs in evaluated strength requirements by using classification society's empirical formula. Therefore, a direct calculation method is needed for evaluating the strength in the flare region in the design stage of hull surface. Inviscid potential-flow based numerical methods are used widely for evaluating wave resistance because of its' robustness and less computational time requirements. However, flow separation, generation of vortex and non-linear wake field occurs in real conditions, which are due to viscous effects. Therefore, improvement is necessary in numerical calculation methods. The possible candidate is the Reynolds averaged Navier-Stokes (RaNS) based computational fluid dynamics method (CFD).

The hydrodynamic load experienced by flare region on the forward part of the hull surface due to ship motions and wave action is termed as bow flare slamming in naval hydrodynamics. In actual operating conditions, bow part of the hull surface lifts completely out of water and hits the free surface. It implies the necessity of studying water entry of 2D sections before proceeding to 3D ship models. Also, the velocity of hull is not constant when it hits the free surface. Therefore, water entry of 2D sections of a 30 deg. wedge and ship is carried out with commercial numerical method called FLUENT® at variable velocity. Water entries of these two sections are carried out because of the availability of their drop test results. Incorporation of velocity variation is carried out by fitting 4th order polynomial as a function of impact time in the experimental velocity profile of the sections and take it as an input velocity in the numerical method. During numerical simulation, velocity profile is updated as each time step. In the simulation domain, the body is held stationary and flow past it is achieved by having a velocity inlet at the bottom of the domain. At the top of the domain, there is a pressure

boundary set at atmospheric pressure to allow outflow of excess air. The sectional geometry is defined as wall boundary condition with no-slip and the far end of the domain is defined as wall boundary conditions with free slip. Geometrical symmetry about the centerline allowed the flow to be simulated in half of the domain. To avoid high skewness of the mesh, multi-block domain is used. Slam loads and local pressure at the measuring points of the experiments along the sectional surface for wedge and ship section at different instances are evaluated and compared with experiment and RaNS based method at constant entry velocity. It is observed that by incorporating velocity variation, well agreement with experiments for local pressure predictions both of wedge and ship sections are achieved than other RaNS based method. Also, comparison of slam loads with experiments for wedge and ship section show that effect of velocity variation for ship section is significant.

For analysis of 3D ship models, in-house RaNS based CFD code named WISDAM-X is used. Improvement and modification of WISDAM-X is carried out to overcome the limitations and applicability of the method in present analysis. As experimental results of slamming for the ship models KCS and SR108 are not available, numerical method is first validated with available experimental results. Validations are made for regular head and oblique waves with heave and pitch free conditions. Reasonably well agreement of computed results with experiments for ship motions motions in both oblique and head waves is achieved. Total drag does not compare well with experiments in head wave conditions. The possible reason is the difference in flexibility of towing the model during experiments and numerical method. Other researchers also compare their numerical methods for regular head wave cases with the same experimental results. Comparison for resistance data show similar manner like present numerical predictions.

Analysis of slamming on the bow flare region is carried out by visualization of flow field. Visualization of numerical results of WISDAM-X for slamming conditions is used. Simulation conditions for slamming events are achieved for a particular ship speed with ratio of wave length to ship length and wave amplitude to ship length in regular head waves for containership model named KCS. The instances for slamming are found. Contour of wave elevation and contour of pressure on hull surface during slamming events are analyzed to find the region on the flare part of the hull surface where slamming occurs with high pressure load. Fig. 1 shows the contour of pressure and its close-up view on the bow flare region during slamming events. Non-dimensional pressure (integrated pressure on the hull surface) value of 2.2 exists in the flare region during slamming events. Pressure is non-dimensionalized based on density of water, square of ship's speed. Flare region of KCS exposed to slamming is in between forward part (F.P.) and station no.9. This region is further divided into sections at an equal interval. Selected particular region and shape of different sections of that region is shown in Fig. 2. In present study, these sections are named as BF1, BF2, BF3, BF4, BF5 and BF6; here BF can be abbreviated as Bow Flare. Convex shaped bulb on the bottom part with low angled concave shape flare exists for section BF1 whereas high angle concave shaped flare angle with no bulb in the bottom part is for section BF6. Other sections in between BF1 and BF6, the proportion of bulb and flare angle is changing in such a way to form a continuous smooth surface.

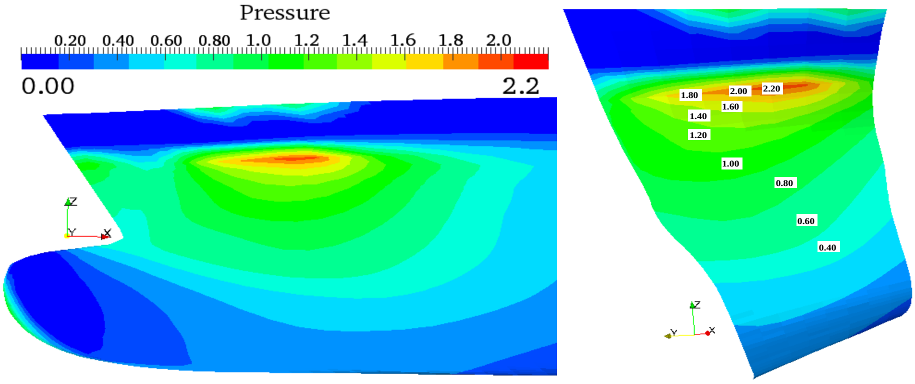

Figure 1: Contour of pressure and its close-up view on the bow flare region during slamming instance for KCS

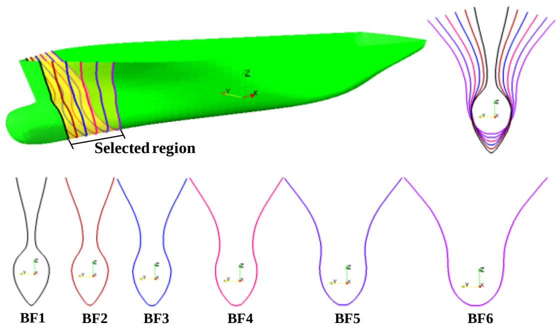

Figure 2: Selected particular region and shape of different sections of that region

Water entry of sections named BF1, BF2 and BF6 are carried out at variable velocity by using commercial finite volume based CFD code FLUENT®. The reason for choosing these three sections is for their sectional shapes as bulb with lower flare, bulb with higher flare and flare without bulb. Numerical results are presented with position of free surface colored by volume of fluid function and velocity vector colored by velocity magnitude at instances. The instances are chosen based on BF1 to analyze the effects of bulb on water entry. The instances are counted initially after the section touching the free surface. The instances are when the flow past the bulb section and its immediate after, when flow is at the flare section and when the section completely goes into water. Fig. 3 shows the location of free surface colored by volume of fluid (VOF) function, contour of velocity magnitude for sections BF1, BF2 and BF6 at time instances 0.025sec.Visualization of numerical results of water entry of sections shows that two times more than that of initial entry velocity (2.453 m/s) is concentrated on bulb and flare for the section with bulb in all the instances considered. Due to presence of bulb in the bottom of the section, the contact surface area with water increases with time after initial impact of surface in the free surface. This increased area displaces additional mass of water, which is called added mass in naval architecture term. Added mass increases the momentum of the flow. Visualization of velocity magnitude contours of the sections at instances for WISDAM-X's computed results in slamming conditions made and shown in Fig. 4 for instance $0.675T_e$, here T_e is one wave period. In this case, 10% more velocity magnitude than that of uniform flow is concentrated on the flare part.

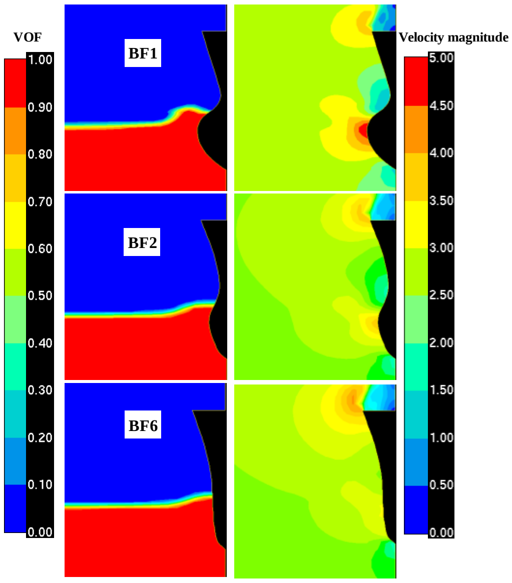

Figure 3: Position of free surface and contour of velocity for sections BF1, BF2 & BF6 from 2D simulations at instance 0.025 sec.

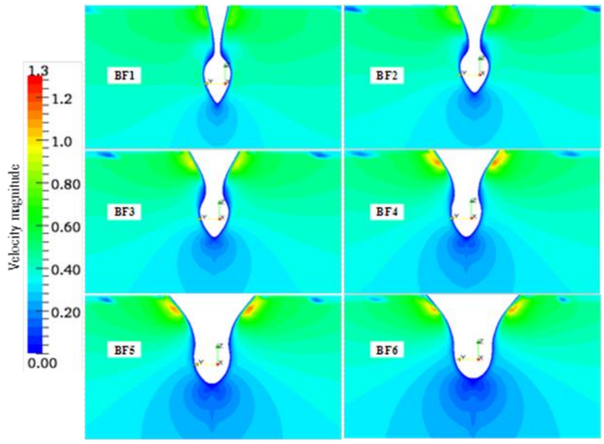

Figure 4: Contour of velocity magnitude for sections BF1, BF2, BF3, BF4, BF5 & BF6 from 3D simulations at instance $0.675T_e$

Three dimensional shape effects on slamming are discussed between container ship models SR108 and KCS. The basic shape difference between SR108 and KCS is that SR108 has large flare (compare to KCS) with small bulb on the lower bottom of forward part of the hull surface and round shaped stern whereas KCS has bulbous bow on the forward part with transom stern. Numerical simulation for SR108 is conducted with the same slamming condition for KCS. Visualization of computed results shows that although SR108 experiences around 45% more pitch motions than that of KCS, no slamming occurs on the flare part of SR108. Amplitude of heave motions for both SR108 and KCS is almost equal.

Three important aspects as mentioned below are noticed in present study;

1. Slamming can happen in regular head waves for hull surface that has the bulbous bow on the forward part
2. Flare region does not experience high pressure in the highest wave elevation
3. Location of high pressure region in the flare part is under the free surface which cannot be visualized in towing tank experiments

Present analysis shows that numerical method WISDAM-X can be used as an effective tool for hull form improvement and numerical results can be used in structural strength analysis procedure especially for the bow flare region on the forward part of the hull surface.