11th International Workshop on Ship and Marine Hydrodynamics
Hamburg, Germany, September 22-25, 2019

Numerical Simulation of Ship Motion and Life Boat Launch using Overset Mesh

Alok Khaware[1,] *, Vinay Kumar Gupta[1], Hemant Punekar[1], Patrick Sharkey[2]

[1]ANSYS Inc. Fluids Development, 411057, Pune, India
[2]ANSYS UK Ltd, Fluids Development, Milton Park, UK

*Alok Khaware, alok.khaware@ansys.com

ABSTRACT

Marine systems need to be designed for performance and safety. Numerical simulations are extensively used to aid this design process. These simulations involve complexities associated with free surface flow modelling along with moving bodies. To overcome some limitations of the traditional dynamic mesh approach, in the present work overset mesh approach has been employed, which provides ease of mesh generation and modelling of mesh motion.

Two common marine hydrodynamic problems are simulated using the Volume of Fluid method (VOF) and overset approach. The first problem focuses on the dynamic response of a container ship at open sea. Accurate prediction of the ship response to the wave plays an important role as it may add resistance altering the speed of the ship or may compromise the structural integrity. The steady-state free surface simulation around the KRISO container ship (KCS) hull is performed where water elevation on the ship and resistance coefficient are validated against experiment. Transient simulation is performed on full-scale KCS hull model where ship interaction with the head wave is captured using a 5th order Stokes wave theory and six-degree of freedom solver. The heave and pitch motion of the ship is validated against the published data.

The second problem focuses on the launching of a lifeboat in the calm water. The lifeboat gets high acceleration due to free fall and the impact point during water entry becomes an important consideration for the safety of passengers. Injury potential of an occupant is directly linked with the acceleration ratio of the boat and the maximum limiting values are recommended by the International Maritime Organization (IMO). Transient simulation for the lifeboat launching is performed where acceleration and boat motion profile are studied in detail.

1 INTRODUCTION

Seakeeping is one of the important areas of research to find a Safe Operating Envelope (SOE) and operating cost of the ship. The damage that can occur at high sea state and the possible transient load is vital information for ship design. Increased resistance in the waves causes more fuel consumption and travel time. Another major concern is also to reduce the greenhouse gas emissions produced by the shipping industry and a strong incentive to achieve the optimum operating performance of ships in waves. When ship interacts with waves, the heave and pitch motions of the ship are the important parameters to focus with respect to the structural integrity of the ship. Resistance coefficient C_T is considered as the measure of performance which is calculated as follows.

$$C_T = \frac{R_T}{\frac{1}{2}\,\rho\, S_{DWL} U^2} \tag{1}$$

Where, R_T is the total resistance, ρ is the density of liquid, S_{DWL} is the wetted surface area on the submerged hull and U is the speed of the ship.

Lifeboat launching from a marine vessel or offshore platform is another concerned area of research in offshore applications. The purpose of a lifeboat is to evacuate people safely during the emergency. Lifeboats are launched from a certain height to avoid the collision with the propulsion system of the marine vessel. A lifeboat should be designed such that it does not get damaged during water entry. Another important factor is that the lifeboat should not be accelerated beyond a limit that is tolerable to the occupants. The injury potential for an occupant in a free-fall lifeboat is evaluated by the acceleration using Square Root Sum of the Squares acceleration (SRSS) method. The IMO recommended limiting value of acceleration in forwarding direction is 15 times of the gravitational acceleration and in vertical downfall and sidewise direction is 7 times of the gravitational acceleration. The Combined Acceleration Ratio (CAR) is calculated depending on the SRSS criterion. The CAR Index less than IMO criteria (IMO, 2003) is found to be safe for the passengers [1].

$$CAR = \sqrt{\left[\frac{g_x}{G_x}\right]^2 + \left[\frac{g_y}{G_y}\right]^2 + \left[\frac{g_z}{G_z}\right]^2} \tag{2}$$

Where, g_x , g_y , and g_z are the concurrent accelarations in the x, y and z seat axis respectively. G_x , G_y and G_z are acceptance limit of acceleration.

In the past, empirical data were used to understand ship hydrodynamics or lifeboat launch. It is very cumbersome and expensive to create a real-life condition for experiments. For lifeboat launch, experiments do not provide details of pressure and velocity distribution around the hull during water entry, subsequent dive and resurfacing of the lifeboat. With the advancement of CFD to solve fluid induced motions coupled with 6DOF solver, it is now possible to predict the motion of the lifeboat with acceptable accuracy and to achieve the detailed analysis of full-scale ships in different challenging sea conditions. The launching conditions for lifeboat can be easily modified with respect to wind speed, water depth, lifeboat launch height, etc. Different types of wave interactions and surrounding conditions can easily be evaluated using CFD.

Traditionally the dynamic remeshing approach was used to solve this kind of problems but it has some drawbacks in controlling good quality mesh in transient moving mesh cases. In the context of free surface flows, the overset method could be a better alternative to dynamic remeshing for ship-wave interactions and lifeboat launches. A refined overset mesh can be strategically positioned to resolve features of interest, including bow waves, wakes, free surface region, etc.

2 OVERSET MESH

Overset method comprises of two or more cell zones overlapping each other. Since different cell zones are not required to be aligned with each other, zonal grids can be generated completely independently. Hole-cutting is the process to deactivate multiple cells at overlapping zones and establish an interface to couple cell-zones to provide a continuous solution. Deactivated cells are called as dead cells. Cells in a zone which provide information to any other overlapping cell zones are termed as donors. Receptors, or interpolation cells receive information from corresponding overlap cell zone donors and vice-versa. In overset interpolation method, receptor cells at the overset interface receive information from multiple donors, which are interpolated as below

$$\phi^h = \sum_{i=0}^{N_d} w^i \phi^i \tag{3}$$

Where, w is the interpolation weights, ϕ is the solution variable and N_d is the number of donors.

A schematic drawing of overlapping component and background zone is shown in Figure 1a. and extracted flow domain from the overset method is shown in Figure 1b.

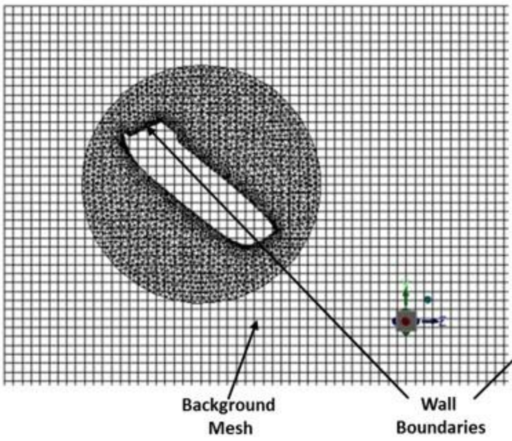

Figure 1a Schematic drawing of overlapping cell zones

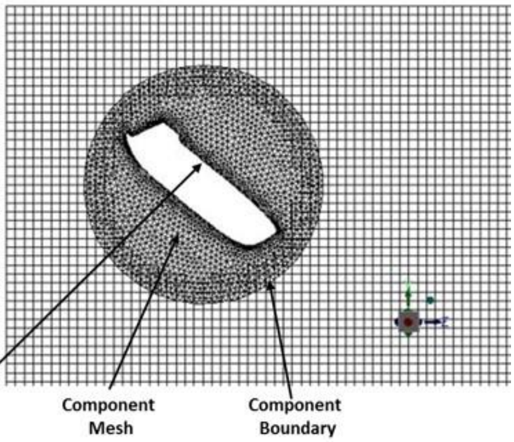

Figure 1b Extracted domain after

As the inter-grid solution coupling is achieved via interpolation in the overlap regions, we obtain a method that is not fully conservative. Figure 2 shows one rectangular receptor cell from Grid-2 overlapping with triangular cells of Grid-1. The number of donors for each receptor is chosen from the neighbourhood of central donor based on either face connectivity or node connectivity. The central donor for a given receptor is the donor cell in the overlapping cell zone that bounds its centroid. Face connected donors are cells which share faces with the central donor as represented by solid lines. Node connected donors are cells which share nodes with the central donor.

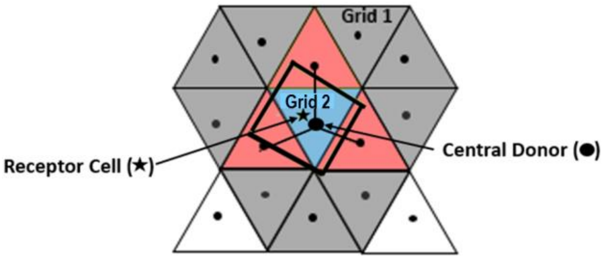

Figure 2: Overlapping grid-1 and grid-2 for overset mesh

In the overset method, interpolation weights are calculated purely based on receptor and donor connectivity. Receptor cell values are updated by the interpolation method followed by computation of reconstruction and viscous gradients. A linear system is built which establishes the mesh connectivity for each cell and its neighbours from same or different grids. The resulting linear system with inter grid coupling is solved simultaneously. After solving the linear system of equations, information on receptor cells are updated from corresponding donors. Information on receptor cell faces is updated from the receptor cell and its neighbours on the same grid using standard discretization procedures [2].

Overset method is in general not conservative, but conservation errors are substantially reduced by keeping the cell size ratio between receptor and donors within a reasonable range, the ideal case would be 1:1. One of the major challenges in overset mesh generation is to deal with orphan cells which are receptors without any valid donors. The presence of orphan cells might make the numerical solution very unstable and could lead to unphysical results. ANSYS Fluent has a numerical treatment to update orphan cells based on neighbour cell data which is active by default. However, it is still advised to have enough mesh resolution to avoid orphan cells, where possible [3].

3 OVERVIEW OF NUMERICAL METHODS

3.1 Governing Equations for CFD Model

The governing equations for CFD are based on conservation of mass, momentum, and energy which are solved using Finite Volume Method (FVM). VOF method is used to capture the interface between the immiscible fluids. If the volume fraction of one fluid in the cell is denoted as α, then α = 0 means that the cell is completely filled with primary fluid; α = 1 means that the cell is completely filled with secondary fluid and $0 < \alpha < 1$ means that the cell is partially filled and contains the interface. Summation of volume fraction for all the fluids should be equal to one

$$\sum_{\alpha} \alpha = 1 \tag{4}$$

Volume fraction equation is given as

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\vec{u}\alpha) = 0 \tag{5}$$

Global continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\vec{u}\rho) = 0 \tag{6}$$

A single momentum equation is solved throughout the domain and the resulting velocity field is shared among the phases

$$\frac{\partial \rho \vec{u}}{\partial t} + \nabla \cdot (\rho \vec{u} \otimes \vec{u}) = -\nabla p + \nabla \cdot \overline{T} + \vec{F}_b \tag{7}$$

Where, $\rho, p, \vec{u}, \overline{T}, \vec{F}_b$, t are represented as density, pressure, velocity vector, stress tensor, body force and time respectively. The properties in the total continuity and momentum equations are volume weighted averaged properties.

3.2 Pressure Based Coupled Solver

The pressure-based coupled solver is very critical to obtain a robust and efficient solution using overset mesh. Coupled solver solves the momentum and pressure-based continuity equations together. The full implicit coupling is achieved through an implicit discretization of pressure gradient terms in the momentum equations, and an implicit discretization of the face mass flux, including the Rhie-Chow pressure dissipation terms.

3.3 Volume Fraction Discretization using Interface Capturing Scheme

In volume fraction equation, face values of volume fraction used in the convection term are discretized using a modified compressive scheme available in ANSYS Fluent which is a second-order reconstruction scheme based on slope limiters [3].

$$\alpha_f = \alpha_d + \beta \nabla \alpha_d . dr \tag{8}$$

Where, α_f is face volume fraction, α_d is donor cell volume fraction, $\nabla \alpha_d$ is donor cell volume fraction gradient, β is slope limiter and dr is position vector between cell to face centroid.

3.4 Six DOF Solver

Six DOF solver calculates the translational and angular motion of the object using forces and moments acting on the object. The governing equation for the translational motion of the centre of gravity is solved for in the inertial coordinate system is

$$\dot{\vec{V}}_G = \frac{1}{m} \sum \vec{f}_G \tag{9}$$

Where $\dot{\vec{V}}_G$ is the translational motion of the centre of gravity, m is the mass and $\vec{f}_G$ is the force vector due to gravity.

The angular motion of the object $\dot{\vec{\omega}}_B$ is computed using body coordinates as

$$\dot{\vec{\omega}}_B = L^{-1}\left(\sum \vec{M}_B - \dot{\vec{\omega}}_B \times L\dot{\vec{\omega}}_B\right) \tag{10}$$

Where L is the inertia tensor, $\vec{M}_B$ is the moment vector of the body and $\dot{\vec{\omega}}_B$ is rigid body angular velocity vector. Moments then transformed from inertial to body coordinates using

$$\vec{M}_B = R\vec{M}_G \tag{11}$$

R is transformation matrix and represents following

$$\begin{bmatrix} C_\theta C_\psi & C_\theta S_\psi & -S_\theta \\ S_\varphi S_\theta C_\psi - C_\varphi S_\psi & S_\varphi S_\theta S_\psi + C_\varphi C_\psi & S_\varphi C_\theta \\ C_\varphi S_\theta C_\psi + S_\varphi S_\psi & C_\varphi S_\theta S_\psi - S_\varphi C_\psi & C_\varphi C_\theta \end{bmatrix} \tag{12}$$

Where $C_x = cos(x)$ and $S_x = sin(x)$. The angles φ, θ and ψ are Euler angles that represents rotations about Z axis (yaw), rotation about Y axis (pitch) and rotation about X axis (roll). Rates are derived by numerical integration; these translational and angular velocities are used to update rigid body positions [3].

4 CASE STUDIES

Four case studies are demonstrated in the present work to validate the overset approach for solving free surface flows. The free surface is modelled using a VOF method available in ANSYS Fluent. A pressure based coupled approach is used to solve flow in all the test cases. The compressive scheme is used for the volume fraction discretization. Interfacial anti-diffusion is applied to capture sharp interfaces. Gradients are calculated using Least square cell-based approach. Body Force Weighted and Second order schemes are used for pressure and momentum respectively whereas First order schemes are used for turbulent quantities. Bounded Second Order time formulation is used for the discretization of transient terms. SST-k omega turbulence model is used for turbulence closure. Six degrees of freedom (6DOF) rigid-body motion solver is applied for the motion of moving objects (more details can be found in ANSYS Theory Guide [2]). Open Channel Flow and Numerical wave tank model are used to provide upstream and downstream boundary conditions. 5th order Stokes wave theory is applied for modelling of non-linear surface gravity waves.

4.1 Steady State Simulation of KCS-Hull in Calm water

The steady-state simulation of KCS hull in calm water is validated in the first case study. A scaled model of KCS Hull with a scaling factor of 31.599 is considered and the relevant information is provided in Table 1. Experimental data about the water elevation at the hull surface is available for KCS hull without radar configuration [4]. The same configuration is modelled in this study using the overset approach on a half-symmetric geometry.

Table 1: KCS hull scaled model parameters

Length between perpendiculars (L_{PP})	7.2786 m
Draft (d)	0.3418 m
Wetted surface area (S_{DWL})	9.4379 m^2
Velocity (U)	2.1962 m/s

A good quality tetrahedral mesh with a skewness less than 0.9 along with 10 inflation layers on hull surface is created for ship component, whereas hexahedral mesh is generated for the background domain. Mesh is refined at the free surface. Various aspects of meshing strategy are highlighted in Figure 3.

The Pseudo-transient approach is used for steady state simulation with a pseudo time step size of 0.005s. Open-channel boundary condition is used for defining free surface level, inlet velocity and hydrostatic profiles at the pressure outlet. Symmetry boundary conditions are applied at top and bottom faces.

The solution is converged well with a residual convergence criterion of 1e-06 along with convergence of drag monitor on the hull. Figure 4 shows the volume fraction of water on the hull surface. It can be observed that interface is captured sharply without any trapped air pockets below the hull. Figure 5 shows the comparison of experimental and numerical results for free surface elevation on the hull. The calculated resistance coefficient (C_T) for the ship hull with no radar configuration is 0.0032667. All the results match well with experimental data.

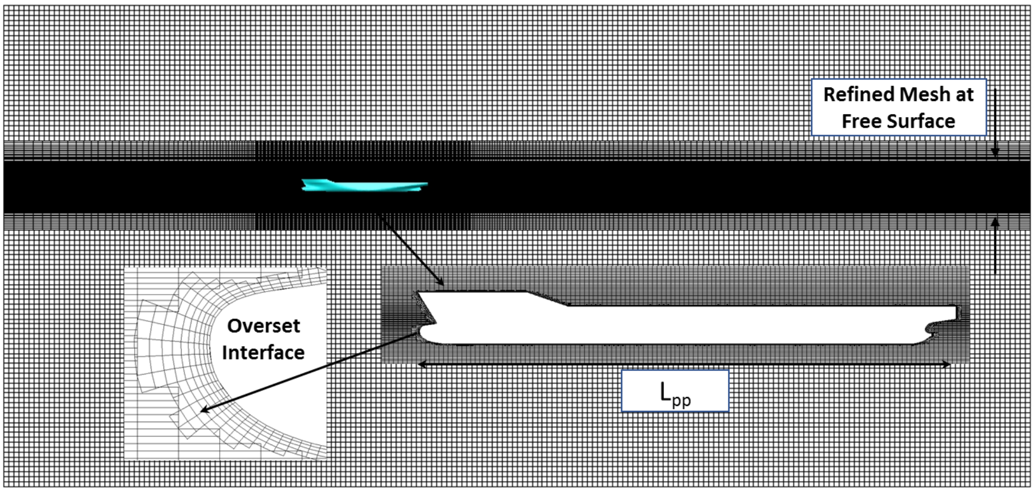

Figure 3: Overset mesh in the domain

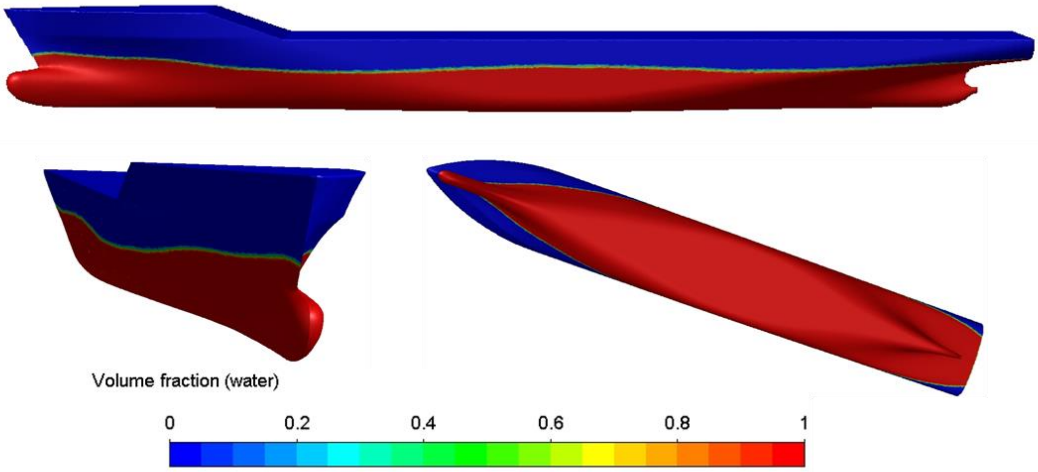

Figure 4: Contour of volume fraction of water on hull surface

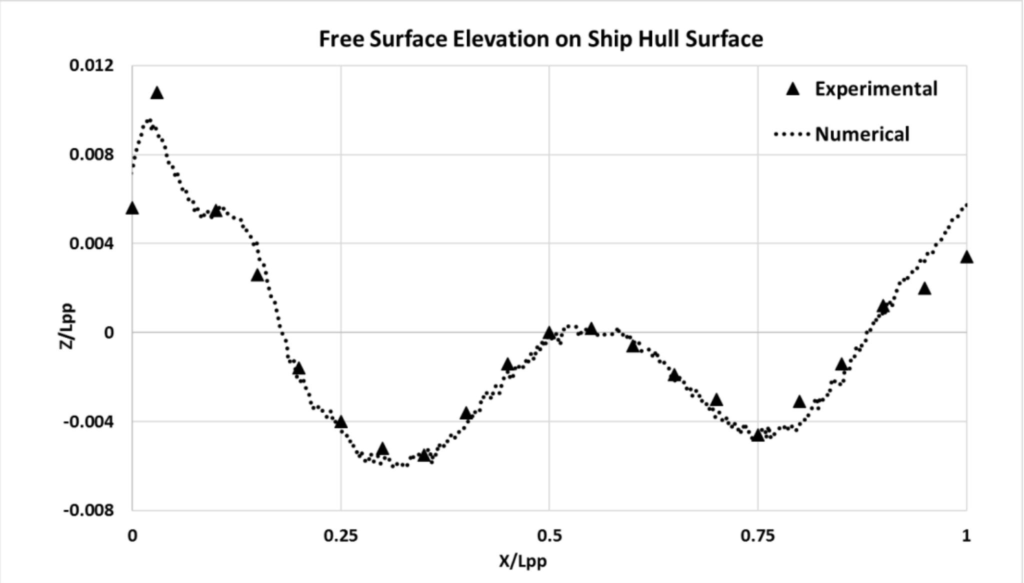

Figure 5: Experimental and numerical results for free surface elevation on the ship hull surface

4.2 Transient Heave Oscillation of a Floating Cylinder

This case study is targeted to validate the accuracy of VOF model and 6 DOF solver with the overset mesh implemented in ANSYS Fluent commercial software. A freely floating cylinder with a given initial condition is undergoing an oscillatory motion, the transient heave decay response is validated with experimental and theoretical data.

This case is modelled based on the experimental work done by Soichi Ito [5]. A cylinder is kept 0.0254 m (1 inch) below the free surface level of water at its initial condition and then allowed to oscillate freely till it reaches the equilibrium. The diameter and length of the cylinder is kept 0.1524 m and 0.2 m respectively. The mass of the cylinder 1.8233 kg is considered such that the cylinder will be half submerged at the equilibrium state. Towing tank dimensions are taken as 20m x 0.2m x 2.4384m. A computational domain of the same dimension is constructed for overset background zone. A component mesh is created around the cylinder. The complete hexahedral mesh is created in background and component zones and mesh independence study is done on three different meshes as mentioned in Table 2.

Table 2: Mesh information for mesh independence study

Mesh Level	Number of Elements at Background	Number of Elements at Component	Total Number of Mesh Elements
Coarse	56 K	72 K	128 K
Medium	125 K	385 K	510 K
Fine	506 K	648 K	1154 K

The case is solved as laminar and ran for 3 seconds of physical time with a time step size of 5e-04 s. This case is also solved with a non-overset layering approach for the comparison purpose. The initial and final position of the cylinder along with the heave decay response curve is shown in Figure 6. The results from the overset approach show a good agreement with theoretical, experimental and non-overset approach results and provide confidence to apply this approach for large marine and offshore applications.

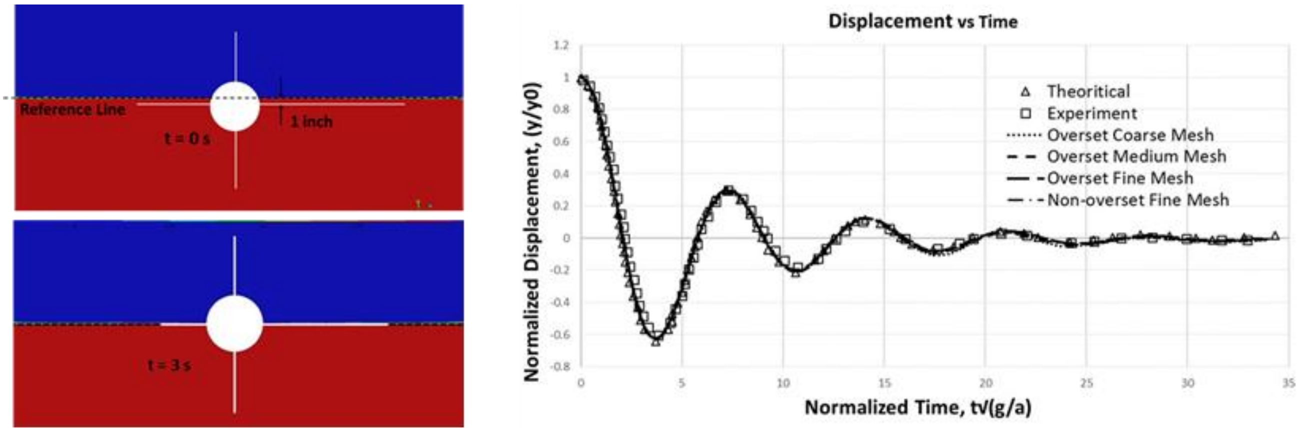

Figure 6: Comparison of transient heave decay with theoretical and experimental

4.3 Transient Simulation of the Full-Scale KCS Hull Motion in Head Waves

This case study targets the transient simulation of full-scale KCS hull in head waves. A non-linear wave with a wave-height (*H*) of 4.424 m, wavelength (*L*) of 264.494 m and time-period (T_e) of 8.097 s interacts with the ship moving with velocity (*U*) of 12.3467 m/s. The important properties of KCS full-scale model are provided in Table 3 [6]. The mesh created for steady-state simulation demonstrated in the first case study is scaled and used for the full-scale transient simulation. Total mesh count for this case is around 11M elements.

Table 3: Properties of full-scale KCS hull

Length of waterline (L_{WL})	232.5 m
Length between the perpendiculars (L_{PP})	230 m
Depth (D)	19 m
Beam at waterline (B_{WL})	32.2 m
Displacement (Δ)	52030 m^3
Vertical centre of gravity from keel	7.28 m
Longitudinal centre of gravity from the aft peak	111.603 m
Moment of inertia (I_{xx}/B)	0.4
Moment of inertia (I_{yy}/L_{PP}, I_{zz}/L_{PP})	0.25
Mass	51936346 kg

Open channel wave boundary condition is used to provide wave inputs. Numerical beach equal to 1.5 times of wavelength is applied in the region adjacent to the pressure outlet for suppressing the numerical reflections propagating upstream. A timestep of $\Delta t = T_e/5000$ s is used with 30 iterations per time step.

Heave and pitch displacements are monitored from the CFD simulation and validated against the published results by Tezdogan et al (2015) [6] as shown in Figure 7 and Figure 8. The heave and pitch motion plots show a close match with the published data after few time periods which may be due to inconsistecies in initialization procedure. Free surface contours are shown in Figure 9 which represent the wave interactions with the ship during 7th time-period cycle. Figure 10 represents the wave pattern around the KCS hull at a time instance of 7th time-period.

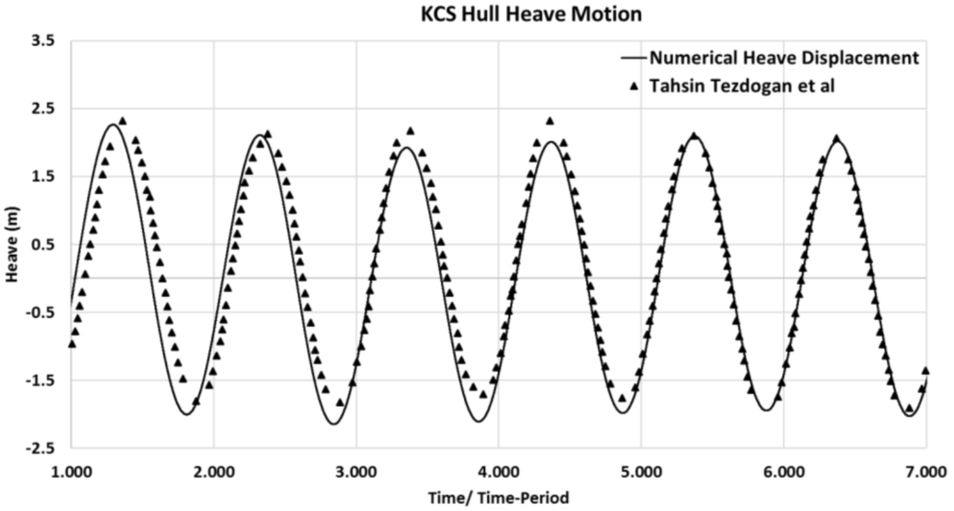

Figure 7: Heave displacement comparison with the published data by Tahsin Tezdogan et al

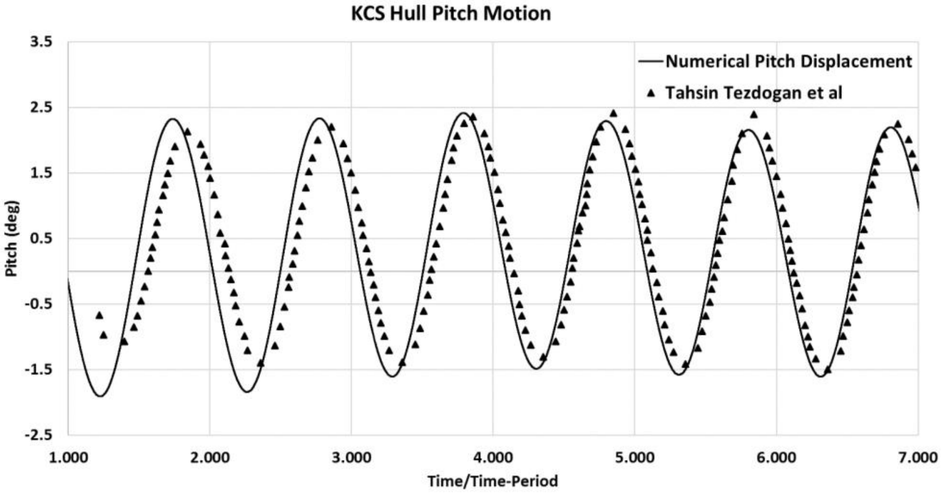

Figure 8: Pitch displacement comparison with the published data by Tahsin Tezdogan et al

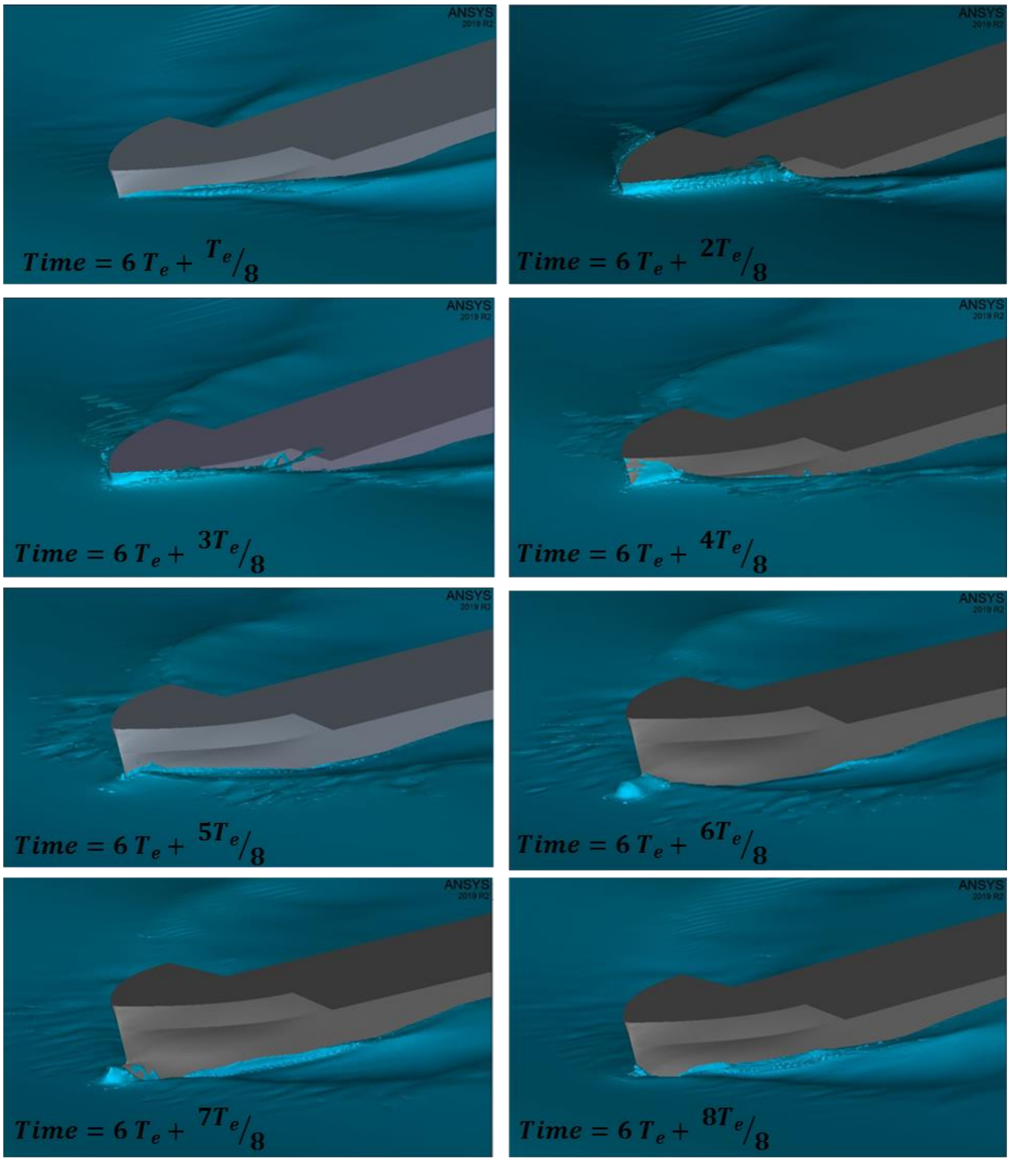

Figure 9 : Free surface contours for 7th time-period (7th T_e)

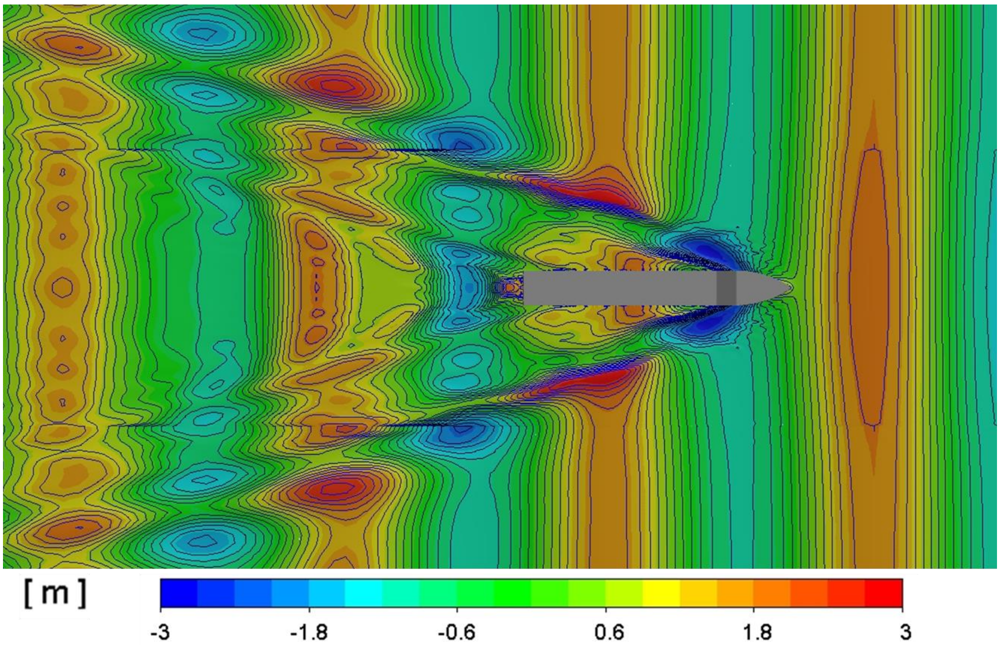

Figure 10 : Wave pattern around the KCS hull at $t = 7T_e$

4.4 Lifeboat Launching

The lifeboat launched from a ramp experiences vertical and angular acceleration as it slides down the ramp. After passing the ramp, lifeboat falls freely in the air before hitting into water. Lifeboat motion and accelerations are generally monitored to optimize the hull shape. In this case study, the lifeboat is launched from 31.4 meters of height with an initial horizontal velocity of 10 m/s, a vertical velocity of 5 m/s and angular velocity of 12 deg/sec. Initial inclination angle of the lifeboat is considered as 39 degrees. Two monitor points are set at front and rear which are shown in Figure 11.

A half symmetry model is prepared for this case. The computational domain is extended 15 meters along the symmetry direction, 75 meters and 95 meters along the vertical and horizontal directions respectively. Around 3.5 million cells are created for overset mesh simulation. A good quality hex mesh is generated for background zone, whereas, tetrahedral mesh with 25 inflation layers on the boat wall surface is prepared for the component zone. Overset component and background mesh along with mesh connectivity at the overset interface are shown in the Figure 11.

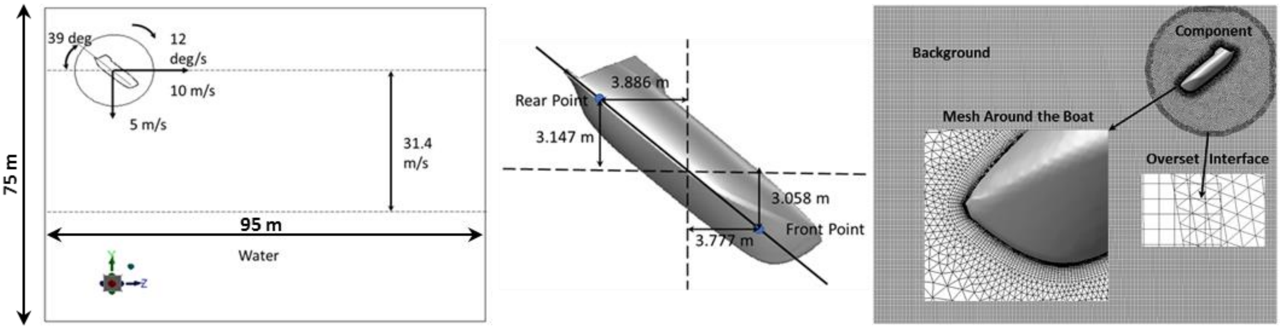

Figure 11: Schematic diagram of lifeboat launch setup, monitoring points and overset mesh

Symmetry boundary conditions are applied at all sides. No slip wall boundary condition is used for lifeboat wall. Water is patched below 31.4 meters from the launch height. 6DOF solver with the implicit update is used to capture the motion of the boat. Rigid body motion settings are defined for lifeboat wall with an initial velocity of -5 m/s in the vertical direction and 10 m/s in the horizontal direction. Component fluid zone remains a passive zone so that it can move along with the lifeboat. Translation along X-axis and rotation along Y and Z-axis are restricted while solving simulation. User defined functions are applied to calculate acceleration and CAR at front and rear points. Water is modelled as a compressible liquid. The same case is solved with the remeshing approach and results are compared with the overset case results.

Figure 12 shows the overset mesh along with volume fraction contour of water during the water entry of the boat.

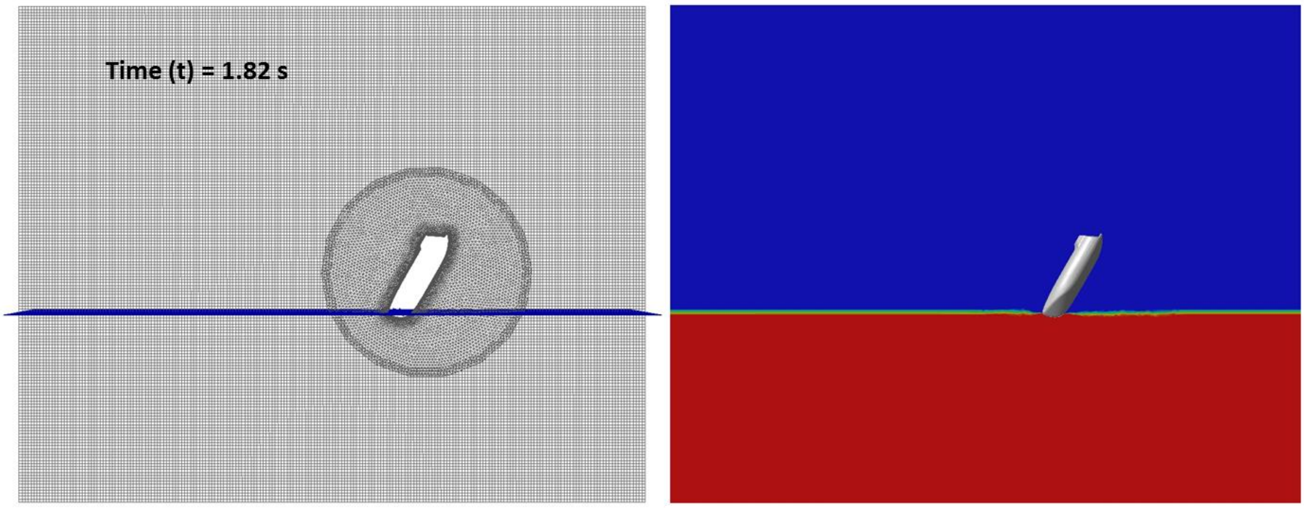

Figure 12: Free fall and water entry of the boat

Different position of the lifeboat during free fall, water entry and resurfacing are presented in Figure 13. Figure 14 presents the comparison of vertical acceleration expressed as multiples of gravitation acceleration at the front monitor point for overset mesh and remeshing cases.

The front point experiences a quick deceleration after the impact with water surface. This initiates a rolling moment and causes a steep acceleration of rear point. The centre of gravity (CG) location, boat mass, and hull shape are critical factors which control the overall maximum acceleration and CAR [8,9]. The lifeboat comes out of the water and re-enters which is called a slamming effect. The slamming can be seen from Figure 13 at t = 5s and t = 6s.

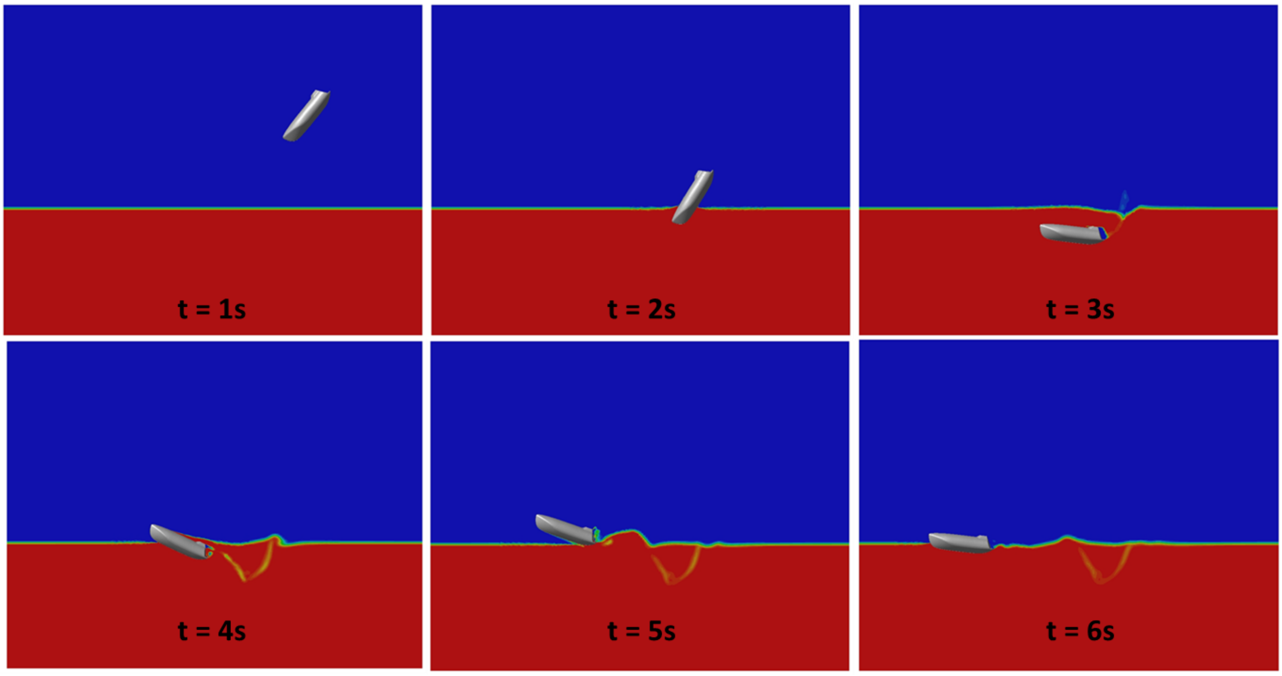

Figure 13: Different location of lifeboat while entering water and resurfacing

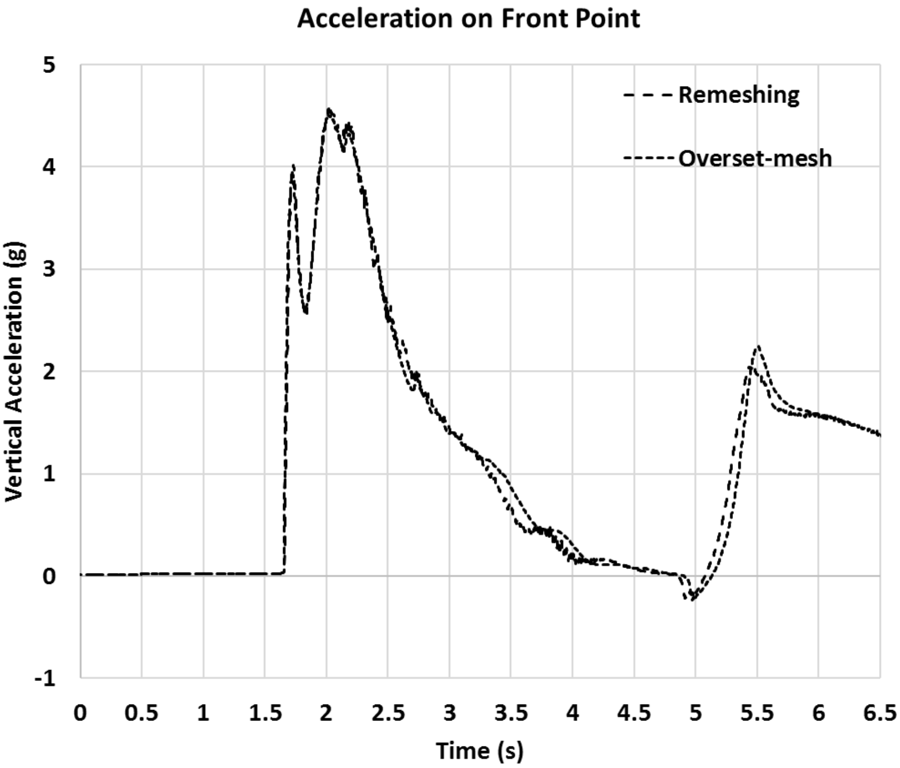

Figure 14: Comparison of vertical acceleration expressed multiples of gravitation acceleration at front monitor point for overset mesh and remeshing cases

5 CONCLUSION

The present paper targets the numerical approach of solving ship hydrodynamics and lifeboat launching using overset mesh and VOF method. Overset methodology facilitates easy creation of meshes for complex geometries with proper resolution and better quality and thus overcomes the shortcomings of traditional dynamic mesh approach. Higher order overset interpolation method helps in achieving accurate results in the overlapping zones. The Coupled approach with implicit volume fraction formulation provides better robustness for overset simulations. The first case study is carried out for the steady-state solution of KCS hull in calm water using static overset mesh and results are validated against experiment. In the second case study, 6DOF rigid-body motion solver is validated for overset mesh by simulating the heave motion of a freely floating circular cylinder. The third case study is targeted on the transient simulation of full-scale KCS hull interacting with non-linear surface gravity waves. The heave and the pitch motion of the ship are validated against published results. The final case study is done for the numerical simulation of lifeboat and overset mesh results are verified with the results obtained from remeshing approach. The results of various validation studies show good agreement with theoretical and experimental data. It can be concluded from the present study that numerical simulation of ship motion and lifeboat launching can be achieved robustly and accurately using overset mesh implemented in ANSYS Fluent.

ACKNOWLEDGEMENTS

We are thankful to Thomas Scheidegger, Srinivas Ramakrishnan and Mohammad Azab for their contribution to continuous development and improvement of overset method in ANSYS Fluent. We would also like to thank Jean-Noel Pederzani for helping in writing user defined functions.

REFERENCES

[1] A. F. Zakki et al "THE DEVELOPMENT OF NEW TYPE FREE-FALL LIFEBOAT USING FLUID STRUCTURE INTERACTION ANALYSIS". Journal of Marine Science and Technology, Vol. 24, No. 3, pp. 575-580 (2016).

[2] ANSYS FLUENT Theory Guide, Release 18.2, ANSYS Inc., 2017

[3] ANSYS FLUENT User Guide, Release 18.2, ANSYS Inc., 2017

[4] V. K. Gupta et al "Development and Application of Interfacial Anti-Diffusion and Poor Mesh Numerics Treatments for Free Surface Flows" 2015 IEEE 22nd International Conference on High performance Computing Workshops, 16-19 Dec 2015.

[5] Soichi Ito, "STUDY OF THE TRANSIENT HEAVE OSCILLATION OF A FLOATING CYLINDER, MASSACHUSETTS INSTITUTE OF TECHNOLOGY" May 1977

[6] Tahsin Tezdogan, et al "Full-scale unsteady RANS CFD simulations of ship behaviour and performance in head seas due to slow steaming". In: Ocean Engineering, 97 (2015) 186-206.

[7] Kim, W.J., Van, D.H. and Kim, D.H., "Measurement of flows around modern commercial ship models", Experiments in Fluids, (2001), Vol. 31, pp 567-578.

[8] Nabila Berchiche, Anders Ostman, Ole Andreas Hermundstad and Svein-Arne Reinholdtsen, "Experimental validation of CFD simulations of free fall lifeboat launches in regular waves". In: Ship Technology Research, (2015), 62:3, 148-158

[9] Hans Jørgen MØRCH et al, "SIMULATION OF LIFEBOAT LAUNCHING UNDER STORM CONDITIONS" 6th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries, SINTEF/NTNU, Trondheim, Norway ,10-12 June 2008