RANSE COMPUTATION OF SLAMMING LOADS ON A PLANING CRAFT

M Kumar and S P. Singh, Indian Register of Shipping, Mumbai, India
V A Subramanian, Indian Institute of Technology, Madras, India

SUMMARY

The paper presents free-surface RANSE computation of a planing craft taking into account dynamic sinkage and trim. The method has been validated with towing tank experiments. The response of the boat to incident waves, vertical acceleration and hydrodynamic impact loads acting on it is also presented. Rigid body motion of the boat and wave dynamics has been implemented in the RANSE solver through user codes. Reynolds Stress model has been used to model the turbulence. Apart from the hull motion and load computation, the paper also provides the validation results for the RANSE computation of water entry problem of a typical wedge section.

NOMENCLATURE

L_P = Projected chine length
B_{PA} = mean breadth over chine
B_{PX} = breadth over chine at transom
B_{PT} = maximum breadth over chine
F_n = Froude Number

1. INTRODUCTION

Ship motions in waves can be predicted using experiments or numerical methods. The numerical methods [1] widely used are based on potential theory, which assumes an irrotational ideal fluid. These methods are considered as fast and robust tools in the design stage as they allow large number of parameters to be analyzed for the purpose of optimization. The motion of simple bodies in waves can be computed using these methods with reasonably good accuracy. However, they are not suitable for flows, where viscous effects or breaking waves play an important role such as roll motion of a ship with bilge keel. Strong nonlinear effects like slamming require more sophisticated methods. The paper presents a method to compute ship motions and slamming loads in waves using a commercial CFD code, Fluent 6.2. The results obtained are compared with experimental results. In order to validate the CFD method to compute slamming loads, simulations are also performed for a typical wedge section [2].

2. NUMERICAL METHODS

The Finite Volume commercial CFD code, Fluent 6.2 has been used to carry out the computations. The numerical approach involves discretizing the spatial domain into finite control volumes using a mesh. The governing equations of mass and momentum are integrated over each control volume, such that the relevant quantity (mass, momentum, energy, etc.) is conserved in a discrete sense for each control volume.

The volume of fluid (VOF) method has been used to capture the free surface. It was originally proposed by Hirt and Nichol [3], who used a finite-difference donor-acceptor scheme to capture the moving free surface interface of a fluid. In the present evolved form, it is widely used to compute the free surface. The location of the two fluids is specified using a volume fraction function, \( \alpha_q \), with \( \alpha_q = 1 \) inside one fluid and \( \alpha_q = 0 \) in the other. The cell for which \( \alpha_q \) lies between 1 and 0 contains the interface between qth fluid and one or more other fluid.

For two phases, \( \rho = \alpha_1 \rho_1 + (1-\alpha_1) \rho_2 \)

For n phases, \( \rho = \sum \alpha_q \rho_q \) and \( \sum \alpha_q = 1 \)

A single momentum equation is solved throughout the domain, replacing density in the original conservation equations with the modified density based on the volume fraction of each phases. The turbulence was modeled using Reynolds Stress model.

2.1 RIGID BODY MOTION

The force on the hull is calculated by integrating the RANSE predicted pressure and frictional shear on the hull. Similarly the moment is calculated by integrating the pressure and frictional shear on the hull along the desired axis. The force and moment acting on the hull are then fed into the solver to compute the translational and angular motion of the center of gravity of the ship hull. The governing equation [4] for the translational motion of the center of gravity is solved in the inertial coordinate system given by Equation 1.

\[
\ddot{v}_c = \frac{1}{m} \sum \dot{f}_c
\]  

Where \( \ddot{v}_c \) is the translational acceleration of the center of gravity about earth fixed co-ordinate system, \( m \) is the mass and \( \dot{f}_c \) is the force vector due to gravity, pressure
and frictional shear. The angular acceleration of the body \( \dot{\omega}_b \) is more easily computed using body coordinates.

\[
\dot{\omega}_b = L^{-1}(\sum \vec{M}_a - \vec{w}_b \times L \vec{w}_b)
\]

(2)

L is the inertial tensor, \( \vec{M}_a \) is the moment vector of the body about the body co-ordinates due to the pressure and frictional shear on the hull, and \( \vec{w}_b \) is the rigid body angular velocity vector. Once the angular and translational accelerations are computed from Eq (1) and (2), the angular and translational velocities are obtained by numerical integration, which are used in the dynamic mesh calculations to update the rigid body orientation.

### 2.2 DOMAIN AND GRID

The details of the ship hull chosen for the numerical simulations have been given in Table 1. The body plan of the hull, Model 4667-1 [5], is shown in Figure 1. The domain used for the numerical computation consists of a rectangular box, which includes both water and air. The ship hull motion has been realized by imposing motion, obtained by solving rigid body motion equations, to a box shaped sub-domain as shown in Figure 2. The volume mesh in the rest of the domain is free to deform. This choice of sub-domain ensured error free grid deformation from moderate to extreme hull movements. The domain is discretized into hexahedral cells using Ansys ICEM-11.0. A meshed domain, with 125398 hexahedral cells, was well able to capture the flow and the motion. The same grid was used in all simulations. A sufficiently refined grid near the hull was ensured. Figure 2 shows the blocking strategy used to generate the hexahedral grid. The surface mesh on the planing hull is shown in Figure 4. Figure 5 shows the mesh on the center plane of the hull.

| LP (m) | 2.438 |
| BPA (m) | 0.487 |
| BPX (m) | 0.596 |
| BPT (m) | 0.381 |
| Mass (kg) | 100.3 |

### 3. PLANING HULL IN CALM WATER

The domain extended to 1.0 L in the front of the hull, 3.0 L behind the hull and 1.5 L each on the both sides of the hull. The distance from the ship to the bottom and top is 1.0 L and 0.7 L respectively. The entire outer boundary, except the outlet and the top, has been modeled as velocity inlet (Figure 2).
Simulations for various Froude number, covering around 0.3 to 2.6, were carried out in calm water. The numerical method allowed the hull to have two degrees of freedom, namely trim and sinkage. Numerically obtained total resistance, trim, sinkage and wetted surface area have been compared, with experiments performed by Eugene and Donald [5], in Figures 6 ~ 9. Numerically computed results compare well with the experimental results.

4. WATER ENTRY OF WEDGE

In order to validate the RANSE based slamming load computation, simulation of the water entry of a wedge section was carried out. The details of the wedge geometry, entrance velocity etc, are given in Zhao et al. [2]. In their experiment, the vertical velocity during the water entry was measured. In the numerical simulation, the vertical velocity was taken as per Figure 10, which is obtained by fitting a polynomial to the curve presented in Zhao et al. [2]. Numerical simulations were carried out for both 2D and 3D cases. The details of the numerical procedure are given in Singh and Kumar [6].

The comparison of 2D and 3D simulations is shown in Figure 11. The results are compared with experimental results of Zhao et al. [2]. It can be observed that 3D result is closer to the experimental result. It is also observed that the 2D simulation over-predicts the force. Hence a 3D simulation is essential.

Figure 12 shows the pressure distribution along the depth of the wedge at time $t = 0.0158$ sec. The pressure is expressed in term of pressure coefficient $C_p$ given as $C_p = \frac{p - p_a}{0.5\rho V^2}$, where $p$ is the absolute pressure, $p_a$ is the atmospheric pressure, $\rho$ is the fluid density and $V$ is the entrance velocity.
5. PLANING HULL IN HEAD WAVE

The inlet velocities are derived from Airy’s wave theory. The velocity components (u, w), for regular plane progressive wave, are given in Eq (3) and (4).

\[ u = U + A \omega e^{kt} \cos(kx - \omega t + \epsilon) \]  
\[ w = A \omega e^{kt} \sin(kx - \omega t + \epsilon) \]  

The wave travels in positive x-direction, z points positive upwards with z = 0 at the undisturbed free surface. A, k, \( \omega \) and \( \epsilon \) are the wave amplitude, the wave number, the encounter frequency and an arbitrary phase angle respectively. The elevation of water is given at the inlet according to Eq (5). Figure 13 shows the wave elevation on a waveprobe located at the midship, 0.8L away from the hull centerline.

\[ \eta = A \cos(kx - \omega t + \epsilon) \]  

**Table 2: Particulars of the Hull used for head sea analysis**

| LP (m) | 1.219 |
| BPA (m) | 0.243 |
| BPX (m) | 0.298 |
| BPT (m) | 0.19 |
| Mass (kg) | 12.5 |

**Table 3: Wave Details**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Fn</td>
<td>0.75</td>
</tr>
<tr>
<td>( \lambda / L )</td>
<td>1.5</td>
</tr>
<tr>
<td>( H / \lambda )</td>
<td>0.05</td>
</tr>
<tr>
<td>Wave Amplitude (A)</td>
<td>0.0457 m</td>
</tr>
<tr>
<td>Encounter Frequency (( \omega ))</td>
<td>14.696</td>
</tr>
</tbody>
</table>

The hull geometry and the domain used for the calm water analysis were scaled down to \( \frac{1}{2} \) for use in the current analysis. This model was scaled down to reduce the simulation time of the transient analysis. The details of the scaled down hull, and the head wave it encounters, have been given in the Table 2 and Table 3 respectively. The motions after a few seconds, when the ship hull oscillates about the equilibrium, are shown in Figure 14.
and 15. The acceleration at a point on the hull in the forward region is plotted in Figure 16. Figure 17 shows the impact pressure at the same location. Figure 18 shows a typical free surface profile around a hull in head waves.

![Figure 16: Acceleration of a point in the forward portion of the hull](image1)

![Figure 17: Impact pressure at a point in the forward portion of the hull](image2)

![Figure 18: Hull in a regular head wave](image3)

### 6. CONCLUSIONS

A computational method to predict motion of a planing craft, and slamming load on it due to its motion in regular head wave, has been presented. Some of the conclusions drawn are as follows:

- It is evident from the experimental comparison of the RANSE predicted resistance, trim and sinkage results that, the body motion coupled RANSE method can predict the calm water resistance and dynamics of a planing craft quite accurately.
- From the wedge water entry results it is evident that the CFD method can accurately predict the impact forces as well.

The method appears to be suitable for impact load computation on a planing craft or similar hulls in waves. More validation studies especially impact load validation in waves, for which experimental data are not widely available, should be carried out.

### 7. REFERENCES


### 8. AUTHORS’ BIOGRAPHIES

**Manoj Kumar** holds the current position of Asst. Ship Surveyor at Indian Register of Shipping, Mumbai, India and working with the ‘ship hydrodynamics’ and ‘noise and vibration’ research teams in the research and development department of the organization. He is a graduate in Naval Architecture and Ocean Engineering from Indian Institute of Technology Madras.

**Prof. V. Anantha Subramanian** teaches at Indian Institute of Technology Madras, India. He is involved with various research projects in the field of ship hydrodynamics. His area of specialization is experimental ship hydrodynamics.

**Sanjay P. Singh** holds the current position of Senior Ship Surveyor at Indian Register of Shipping, Mumbai, India and heads the ‘ship hydrodynamics’ research team in the department. He is also involved with development of various in-house ship hydrodynamics softwares.