Verification and Validation for the Resistance of a KRISO Container Ship in Calm Water

Anzheng Yu, Decheng Wan*, Gang Chen

State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai, China

*Corresponding author

ABSTRACT

The verification and validation (V&V) of the total resistance for a 7m-long KCS model is presented in this paper. In-house hydrodynamic solver naoe-FOAM-SJTU is employed to solve the Reynolds-averaged Navier-Stokes (RANS) equations for unsteady turbulent flows around the ship model. The Volume of Fluid (VOF) is used to capture the free surface. Three sets of systematically refined grids are used for the grid convergence study and the V&V study. The overall results show great correspondence with the experimental data. The verification of the total resistance follows the ITTC recommended procedure and the validation is based on the ASME standard. The time step uncertainty is evaluated on the coarse mesh and the results show that time step uncertainty is insignificant in this study. Besides, although the validation is not achieved in this paper, the comparison error is very close to the validation uncertainty and hence more discussions are required in subsequent work.

KEY WORDS: verification and validation; KCS; naoe-FOAM-SJTU solver; uncertainty.

INTRODUCTION

With the great development of computer technology and facilities in the past decades, Computational Fluid Dynamics (CFD) has become a powerful tool in scientific research and engineering applications. In ship hydrodynamics, CFD has been applied to performance prediction and ship design. Although Experimental Fluid Dynamics (EFD) is still the most reliable and accurate way to predict ship performance, the requirement of facilities and high cost make it a limitation. Compared with the model test, CFD is obviously a more convenient and low-cost tool, and in addition, it can provide insight into the local flow characteristics which is useful for the ship design.

However, because of the discretization, iteration and machine precision, there is an error between the numerical result of CFD and the actual result. The difference can also be expressed as uncertainty, an interval within which the error probably falls. The magnitude of error or uncertainty can be used as a criterion of the numerical result. Currently, verification and validation (V&V) aim to establish the credibility of CFD and evaluate the numerical and modeling errors. As Roache (1998) proposed, verification is the process of determining whether solving the equations correctly as validation is the process of determining whether solving the correct equations, which means that verification evaluates the numerical uncertainty whereas validation deals with the modeling uncertainty. In the process of verification, the source of numerical error mainly comes from the iteration and discretization, of which the discretization error is dominant. By contrast, the validation determines the degree of accuracy to which the numerical model describes the real physical problem combined with the EFD data.

The standardization of verification and validation has been a long time. Almost all proposed V&V procedures are based on Richardson Extrapolation (RE) (Richardson, 1911). Roache (1998) put forward a method called Grid Convergence Index (GCI) to estimate the numerical uncertainty, and as the first standardized V&V procedure, it lay the foundation for subsequent development. Stern et al. (2001a, 2001b) introduced an uncertainty analysis method via a correction factor to guarantee the credibility of uncertainty assessment, and formed the foundation of the International Towing Tank Conference (ITTC) uncertainty analysis procedures (ITTC, 2002). Eça and Hoekstra (2014) recommended a Least-Squares Root approach based on RE and GCI. This method employs the least-square sense to take data scatter into account. Based on the previous work by Stern et al. (2001a, 2001b), Xing and Stern (2010) developed the Factors of Safety (FS) method and provided statistical evidence that the uncertainty can achieve 95% confidence level.

In ship hydrodynamics, Zou et al. (2010) evaluated the numerical error/uncertainty for a tank without appendages manoeuvring at different drift angles and water depths, and the Validation is conducted by comparison with the experimental data. Zhang (2010) used the procedures proposed by ITTC to verify and validate resistance and wave profile of a modern container ship KCS without propeller.
Numerical simulations are conducted in the ship hydrodynamics CFD solver, naoe-FOAM-SJTU. In present work, the CFD code solves the Reynolds-averaged Navier-Stokes (RANS) equations for unsteady turbulent flows around the ship models. The blended k-ε, k-ω shear stress transport (SST) turbulence model is used to simulate the turbulent flow (Menter et al., 2003). Volume of fluid (VOF) with bounded compression technique (Weller, 2008) method is adopted to deal with the air-water interface. The computational domain is discretized by hexahedral unstructured grid and the finite volume method (FVM) is used to transform RANS and VOF equations into computational space. The PISO-SIMPLE (PIMPLE) algorithm is applied to solving the pressure-velocity coupling equations.

**Governing Equations**

The incompressible RANS equations can be expressed as follows:

\[ \nabla \cdot \mathbf{U} = 0 \]  
\[ \frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} - \mathbf{U}_g) = \nabla \cdot (\mu \mathbf{U}) + f_{\text{eff}} \]  
\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \]  
\[ \frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho \mathbf{U} - \mathbf{U}_g) \alpha = \nabla \cdot (\rho U_\alpha) \]  
\[ \frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho \mathbf{U} - \mathbf{U}_g) \alpha = \nabla \cdot (\rho U_\alpha) \]  
where \( \mathbf{U} \) is fluid velocity and \( \mathbf{U}_g \) is the grid velocity; \( p_d \) means the dynamic pressure and \( \rho \) represents the mixture density; \( g \) is the gravity acceleration; \( \mu_{\text{eff}} \) is the effective dynamic viscosity and can be expressed as \( \rho (v + v_i) \), in which \( v \) is the kinetic viscosity whereas \( v_i \) is the eddy viscosity. \( f_{\text{eff}} \) is the surface tension term and set to zero in current work.

**Volume of Fluid Method**

The VOF method with artificial compression is employed to capture the free surface. The transport equation can be written as follow:

\[ \frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{U}) \]  
\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \]  
where \( \alpha \) is the volume fraction, indicating the relative proportion of fluid in each cell, and its value is always falls between 0 and 1:

\[ \alpha = \begin{cases} 0 & \text{air} \\ 0 < \alpha < 1 & \text{interface} \\ 1 & \text{water} \end{cases} \]  

In Eq. 3, \( \mathbf{U}_r \) is the velocity field used to compress the interface, and it only works on the interface where \((1 - \alpha)\alpha \) is not equal to zero.

**Test Conditions**

The present work follows case 2.1 of the Tokyo 2015 CFD workshop and chooses the speed at 2.196 m/s (Fr=0.26) to perform the numerical simulation and verification. The available experimental data can be used for CFD validation. The model test was conducted in calm water and the ship model is free to heave and pitch. The gravity acceleration is 9.81 m/s² and the water density is 999.5 kg/m³. More details can be found at the website of T2015 CFD workshop.
Mesh generation

Unstructured and hexahedral mesh applied in present study is generated by HEXPRESS. Three sets of systematically refined grids with a refinement ratio of $\sqrt{2}$ are used and the refinement is done by splitting the background mesh. The computational domain is shown in Fig. 2. The origin of the coordinates is set at the intersection of the waterline and the hull. The direction of x-axis is from bow to stern and the y-axis is positive to starboard when z-axis points upward. The range of computational domain in three directions are: \(-1.0L_{pp}<x<4.0L_{pp}\), \(0<y<1.5L_{pp}\), \(-1.0L_{pp}<z<0.5L_{pp}\).

Fig. 2 Computational domain

Three refinement boxes are set to accurately capture the complex surface of KCS. Grids around the free surface are also refined with a refinement level of four in the z direction. The hull is refined at level four where the rudder refinement is set to five. The overall view of grid distribution is shown in Fig. 3 and the mesh size ratio is 2 between adjacent level. In addition, to ensure that the grids closest to the solid surface meet the requirements, the y+ values of three sets of meshes are set to 40, 56.57 and 80 respectively. The details of the mesh generation are list at Table 2.

Table 2. Details of mesh generation

<table>
<thead>
<tr>
<th>Grids sets</th>
<th>Background Mesh</th>
<th>y+</th>
<th>Number of Grids</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>200×60×60</td>
<td>40</td>
<td>3.799M</td>
</tr>
<tr>
<td>2</td>
<td>142×42×42</td>
<td>56.57</td>
<td>1.690M</td>
</tr>
<tr>
<td>3</td>
<td>100×30×30</td>
<td>80</td>
<td>0.776M</td>
</tr>
</tbody>
</table>

Convergence study

The results of grid convergence study of the total resistance coefficient \(C_T\), sinkage and trim are summarized in Table 3. In contrast with the experimental data, the error on the fine mesh is 2.147% and the error on the coarse mesh is 3.062%, indicating that present grid sets can accurately model the ship resistance. The differences of sinkage and trim compared with the model test data are blow 2.08%, 5.36% respectively. The above results show that present numerical methods and grid distribution can accurately calculate the overall quantiative data of ship hydrodynamics.

Table 3. Grid convergence of \(C_T\), sinkage and trim.

<table>
<thead>
<tr>
<th>Grids sets</th>
<th>(C_T\times10^3)</th>
<th>Sinkage×10²(m)</th>
<th>Trim(deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.791</td>
<td>-1.388</td>
<td>-0.173</td>
</tr>
<tr>
<td>2</td>
<td>3.804</td>
<td>-1.365</td>
<td>-0.176</td>
</tr>
<tr>
<td>3</td>
<td>3.825</td>
<td>-1.390</td>
<td>-0.178</td>
</tr>
<tr>
<td>EFD</td>
<td>3.711</td>
<td>-1.394</td>
<td>-0.169</td>
</tr>
<tr>
<td>Maximum error</td>
<td>2.147%</td>
<td>2.08%</td>
<td>5.36%</td>
</tr>
</tbody>
</table>

Fig. 4 shows the wave profile of three longitudinal cuts (\(y/L=0.0741, 0.1509, 0.4224\), \(L\) is the length between perpendiculars) under different grid sets. Results from three grid sets are almost identical and all of the cuts exhibit a good correspondence with the experimental data, which is acquired from the assessment of the Gothenburg 2010 workshop (Larsson and Stern, 2014). Besides, the closest cut (\(y/L=0.0741\)) is the best fit compared with the model test data. The cut of \(y/L=0.1509\) also shows good consistency whereas a small deterioration of the fits is found at \(y/L=0.4224\) after \(x/L=1.4\). The above results indicate that present grid density and the VOF method can accurately capture the free surface.

Fig. 4
The wave pattern is compared with other early data provided by Kim et al. (2001). In this model test, the ship model is fixed at the waterline and no rudder is installed. Considering that the small sinkage and trim at Fr=0.26, the model test data can be a reference. Fig. 5 shows the wave profile on the hull surface of model test data and different grid sets. All numerical results fit well with experimental and the difference between three grid sets are negligible. The first crest is located at the bow with a height around 1.1% of the ship length and the first trough, by contrast, can be seen at around x/L=0.3 with a height under -0.5% of the ship length. The wave elevation near the bow is steeper than in the stern region and this can be partly attributed to the flat overhang of KCS.

Fig. 6 shows the global view of the wave pattern of numerical results and EFD data. The Kelvin angle can be observed in the wave system. The divergent wave components and transverse wave components are accurately simulated on three grid sets before x/L=1.6 compared with model test data. The free surface after x/L=1.6 is not captured well in present work and the situation is improved as mesh refined. In general, the overall results of CFD show that present numerical methods and grid distribution can accurately simulate the flow field of KCS model.

Fig. 5 Wave profile on hull surface

Fig. 6 Wave pattern of KCS at Fr=0.26

V&V STUDY OF TOTAL RESISTANCE

Verification of Total Resistance

In general, verification is the process of evaluating the numerical error/uncertainty. The numerical error in present work can be expressed as the summation of the grid error, the time step error and the iterative error.

Time step convergence.

In order to evaluate the time step uncertainty, systematically time steps with a refinement ratio 2 are employed. Three time steps (Δt=0.0005s, 0.001s, 0.002s) are chosen and carried out on the coarse mesh. Results of Cr, sinkage and trim show that the time step has little effect on the
simulation. Hence the uncertainty of the time step is ignorable and the subsequent V&V study is based on the medium time step $\Delta t=0.001s$.

**Iterative convergence.**

Iterative convergence study follows Sakamoto et al. (2012) and is assessed through the running mean of the time history of $C_T$. The expression is shown in Eq. 1 in which $RM_{max}$ and $RM_{min}$ is the maximum and minimum value of the last oscillation of the running mean time history. Table 4 shows the iterative uncertainties $U_i$ on different grid sets and the maximum $U_i$ is 0.663% of the EFD data. Therefore, the iterative uncertainty is non-negligible and should be considered in the numerical uncertainty.

$$U_i = \frac{RM_{max} - RM_{min}}{2}$$  \hspace{1cm} (5)

Table 4. Iterative uncertainties $U_i$ on different grid sets.

<table>
<thead>
<tr>
<th>Grids sets</th>
<th>1</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_i%D$</td>
<td>0.663</td>
<td>0.637</td>
<td>0.529</td>
</tr>
</tbody>
</table>

$D$ is EFD data.

**Grid convergence and verification study.**

Verification study of the total resistance follows the ITTC recommended procedures (ITTC, 2002) and the focus is the assessment of the grid uncertainty. The convergence ratio $R_g$ can be written as:

$$R_g = \frac{\varepsilon_{S_1}}{\varepsilon_{S_2}}$$  \hspace{1cm} (6)

where $\varepsilon_{S_1}$ and $\varepsilon_{S_2}$ are the solutions on the fine, middle and coarse mesh, respectively. Three convergence situations are possible according to ITTC (2002):

1. Monotonic convergence: $0 < R_g < 1$
2. Oscillatory convergence: $R_g < 0$
3. Divergence: $R_g > 0$

The value of $R_g$ in this study is 0.609 and meets the criteria (1). Based on the Richardson Extrapolation (RE), the error can be expressed as follows:

$$\delta_{RE} = \frac{\varepsilon_{S_2}}{\varepsilon_{S_1}^G - 1}$$  \hspace{1cm} (7)

where the $\varepsilon_{S_1}^G$ is the grid refinement ratio $\sqrt{2}$ and $P_g$ is the observed order of accuracy of present work, the expression of $P_g$ is:

$$P_g = \frac{\ln(\varepsilon_{S_2} / \varepsilon_{S_1})}{\ln(\varepsilon_{S_1}^G)}$$  \hspace{1cm} (8)

Besides, the correction factor can be expressed as:

$$C_g = \frac{\varepsilon_{S_1}^G - 1}{\varepsilon_{S_1}^G - 1}$$  \hspace{1cm} (9)

where the $P_{\text{Gest}}$ is an estimate for the limiting order of accuracy as the spacing size goes to zero and its value is 2 in this work. Finally, the grid uncertainty can be written as follows:

$$U_g = \begin{cases} 9.6(1 - C_g)^2 + 1.1|\delta_{RE}|, & |l - C_g| < 0.125 \\ 2|l - C_g| + 1|\delta_{RE}|, & |l - C_g| \geq 0.125 \end{cases}$$  \hspace{1cm} (10)

Table 5 shows the values of $R_g$, $P_g$, $\delta_{RE}$, $C_g$ and $U_g$. The observed order of accuracy $P_g$ is 1.431 and is very close to the theoretical order of accuracy $P_{\text{Gest}}$. The error estimated by RE is below 1% of $S_1$, indicating that the current numerical simulation is well convergent. The correction factor $C_g$ is 0.642 and according to the ITTC procedures, the grid uncertainty is 0.926% of the experimental data.

Table 5. Values of $R_g$, $P_g$, $\delta_{RE}$, $C_g$ and $U_g$.

<table>
<thead>
<tr>
<th>$R_g$</th>
<th>$P_g$</th>
<th>$\delta_{RE}%S_1$</th>
<th>$C_g$</th>
<th>$U_g%D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.609</td>
<td>1.431</td>
<td>0.540</td>
<td>0.642</td>
<td>0.926</td>
</tr>
</tbody>
</table>

$S_1$ is the solution on the fine mesh and $D$ is EFD data.

Based on the iterative uncertainty $U_i$ and the grid uncertainty $U_g$, the numerical uncertainty can be written as:

$$U_{SN} = \sqrt{U_{SN}^2 + U_g^2}$$  \hspace{1cm} (11)

As a result, the numerical uncertainty $U_{SN}$ is 1.139% of the EFD data.

**Validation of Total Resistance**

The validation is the process of evaluating the model error/uncertainty. The present work follows the validation standard provided by ASME (2009). The method is summarized in Fig. 7, in which the source of the uncertainty is presented. Two items are used in the standard: the validation uncertainty $U_{val}$ and the comparison error $E$, they can be written as follow:

$$E = S - D$$  \hspace{1cm} (12)

$$U_{val} = \sqrt{U_{SN}^2 + U_D^2}$$  \hspace{1cm} (13)

where $S$ is the numerical solution, $D$ is the EFD data, $U_{SN}$ is the numerical uncertainty and $U_D$ is the experimental uncertainty (the simulation input error is ignored in this study). The validation procedure of ASME is a combination of the numerical simulation and the model test, which aims to evaluate the modeling error. If $|E| \leq U_{val}$, the validation is achieved and the modeling error is less than the “noise level” caused by CFD and EFD; and if $|E| > U_{val}$, the comparison error is above the “noise level” and the sigh and magnitude of $E$ is valuable to estimate the modeling error from the uncertainty standpoint; whereas if $|E| >> U_{val}$, then the modeling error is the leading role of the comparison error and the validation fails in this level.

Table 6 shows the results of the validation in this study. Combined with
the numerical uncertainty $U_{SN}$ and the model test uncertainty $U_0$, the $U_{val}$ is 1.516% of the EFD data. The comparison error $E$ is based on the solution on the fine mesh and is 2.147% of the EFD data. Obviously the comparison error is above the validation uncertainty, but the magnitude of $E$ is very close to $U_{val}$ and hence further discussions for the numerical model are needed in subsequent work.

(1) In the grid convergence study, the results of the total resistance coefficients, sinkage and trim on three grid sets are compared with the experimental data and all errors are acceptable. The wave profiles of three longitudinal cuts ($y/L=0.0741, 0.1509$ and 0.4224) exhibit a good correspondence with the experimental data. Besides, the wave height on the hull surface and the wave patterns are compared with the model test data. To sum up, the overall results of CFD show that present numerical methods and grid distribution can accurately simulate the flow field of KCS model.

(2) In the verification study, the time step uncertainty is evaluated on the coarse mesh and the result shows that the time step uncertainty is insignificant in current study. In addition, the iterative error is found non-negligible with a maximum value of 0.663% of the EFD data. The grid uncertainty follows the ITTC procedures and the observed order of accuracy $P_G$ is 1.431, which is very close to the theoretical value.

(3) In the validation study, the comparison error and the validation uncertainty are combined to assess the modeling error. The validation uncertainty based on the numerical uncertainty and the experimental uncertainty is 1.516% of the EFD data whereas the comparison error is 2.147% of the EFD data. According to the standard, the comparison error is greater than the validation uncertainty and the validation in this level is not achieved. However, the magnitude of $E$ is very close to the $U_{val}$ and therefore more discussions are required in latter work.

The future work will focus on the verification and validation of the resistance of KCS model under different speeds. In addition, the effect of the mesh domain, turbulence model and the boundary conditions on the V&V study will be investigated.

ACKNOWLEDGEMENTS

This work is supported by the National Natural Science Foundation of China (51879159, 51490675, 11432009, 51579145), Chang Jiang Scholars Program (T2014099), Shanghai Excellent Academic Leaders Program (17XD1402300), Program for Professor of Special Appointment (Eastern Scholar) at Shanghai Institutions of Higher Learning (2013022), Innovative Special Project of Numerical Tank of Ministry of Industry and Information Technology of China (2016-23/09) and Lloyd’s Register Foundation for doctoral student, to which the authors are most grateful.

REFERENCES


Workshop,” Springer.