VALIDATION AND VERIFICATION OF HULL RESISTANCE COMPONENTS USING A COMMERCIAL CFD CODE

C.A. Perez G, University of Southampton, UK. Universidad Pontificia Bolivariana, Colombia,
M. Tan and **P.A. Wilson** University of Southampton, UK.

SUMMARY

A mathematically defined Wigley hull form is used to investigate the application of a commercial CFD code in prediction of the total resistance and its components from tangential and normal forces on the hull wetted surface. The computed resistance and wave profiles alongside the hull were compared with experimental observations for six different Froude numbers to validate the simulations. The effects of grids, domain size and turbulence models were studied. A statistical hypothesis test was carried on these resistance data in order to determine the most suitable turbulence model for the lowest and highest Froude numbers.

1. INTRODUCTION

The resistance of a hull is a consequence of air and water forces acting against movement of the vessel. For this reason its determination is an important issue regarding the propulsion and ways to provide it. The water will have the major contribution to the resistance of a hull, unless there are strong winds. As a result, predictions on resistance are a good way to know how the energy is spent.

Traditional methods to predict resistance on real hulls are based on towing tank models running at corresponding Froude numbers and then scaling results taking into account a friction line for the respective Reynolds number. Advantages of these methods are the knowledge and experience acquired through the years that make results reasonably trustworthy. Disadvantages are the associated cost and the limitation on the availability of physical tanks and models for every single design. This has been one of the motivations of attempting to predict hull behaviours using computational tools, for this specific case, resistance. Although computations started in earlier 60's solving the simplified boundary layers equations[1], followed by methods based on potential flow theories and those to solve Reynolds Navier-Stoke Equations in the last two decades[2], experimental data are still required to validate computational results.

In order to obtain accurate results even in steady state simulations, the problem needs to be set-up carefully and this includes having sufficient nodes within the boundary layer, correct mesh for high gradient zones and suitable time step sizes. Comprehensive efforts have been made to the procedures to verify and validate computational data however there is still a lack of consensus of suitable techniques[3].

In the present study the commercial CFD code used is ANSYS-CFX 11.0 which adopts a false time step or pseudo-time step to solve equations as a means of under relaxation. The under relaxation is necessary to stabilize some iterative processes of obtaining steady state solutions[4]. It could be said that smaller physical time steps are more robust than larger ones[5]. Nonetheless, convergence will require more CPU time.

2. CFD SIMULATIONS

The Wigley hull was selected as a benchmark to gain understanding of free surface simulations using CFX. The hull was generated by a three variables parametric function in ANASYS ICEM and it is parabolic under the waterline and extends up vertically. Equation 1 describes the underwater hull shape and a sketch is given in the Figure 1.

$$y = \frac{B}{2}\left[1-(2x/L)^2\right]\left[1-(z/T)^2\right] \qquad (1)$$

In equation 1 x, y and z are the longitudinal, transverse and vertical ordinates of the hull surface and L,B and T are the length, the breadth and the draught of the hull, respectively. For this case L was chosen as 1m, B=0.10 m and T=0.0625m.

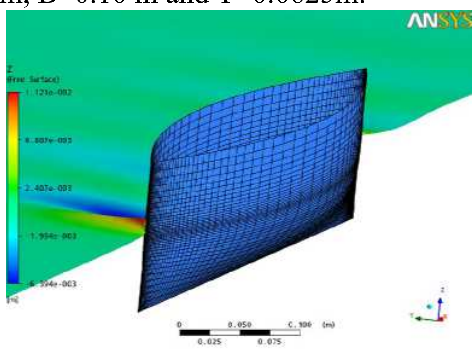

Figure 1. Wigley Hull.

A physical domain with water and air at standard conditions was specified and a homogeneous coupled Volume of Fluid model was selected as it is recommended for free surface flows where the free surface is well defined over the entire domain[5]. An homogeneous model allows two different phases when the interface is distinct and well defined everywhere, as it is the case of hulls riding on a free surface without breaking waves. For this initial simulation the k-ε turbulence model was used. The boundary conditions were imposed as follows, *Inflow*: normal free stream, *outflow*: hydrostatic pressure, *Top*: opening pressure, *Midplane*: symmetry, Side and *Bottom*: free slip. The Figure 2 shows the initial computational domain of 5m long 3m wide and 0.625m deep with 234,320 elements. Table 1 shows the results of the longitudinal components of the tangential and normal forces acting on the hull and experimental data from[6]. Also, figure 3 shows hull wave profiles in terms of non dimensional wave height compared with the data from same source[6].

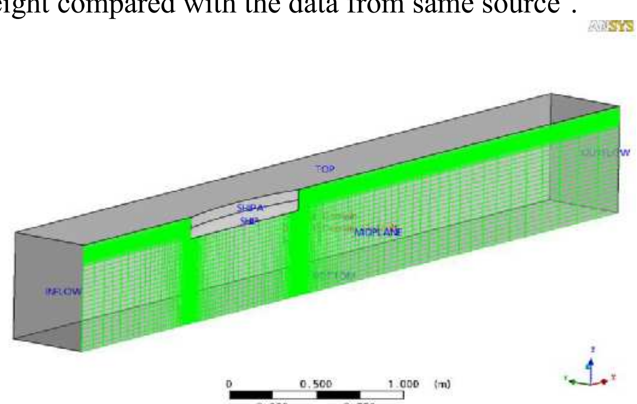

Figure 2. Initial Computational Domain.

Table 1. Computational and experimental resistance coefficients ($\times 10^3$)[6].

Froude Number	0.250	0.267	0.289	0.316	0.354	0.408
Ctang $_x$	4.77	4.72	4.65	4.58	4.47	4.34
Cnormal $_x$	1.03	1.12	1.47	1.75	1.72	2.69
Ctotal	5.80	5.84	6.12	6.33	6.19	7.03
Ctotal Exp	5.92	5.84	6.16	6.32	6.12	6.91
%Error	2.0%	0.1%	0.7%	0.2%	1.2%	1.7%

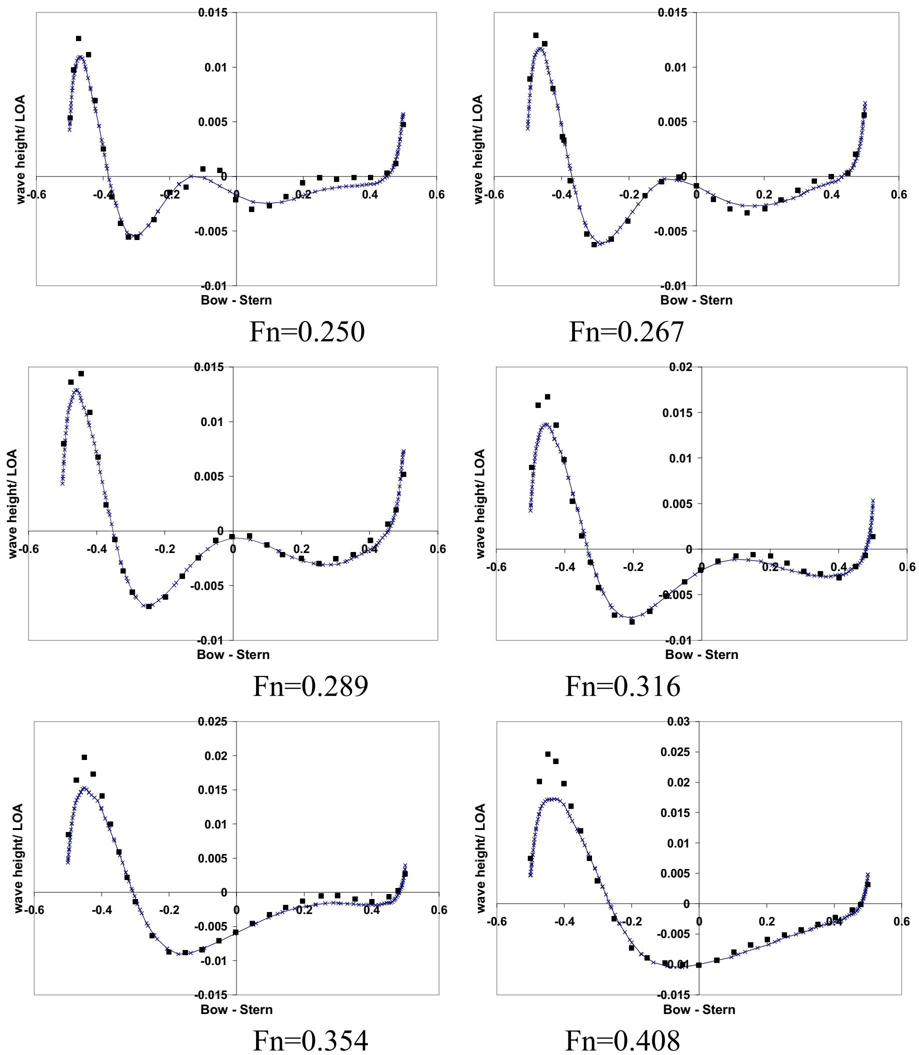

Figure 3. Computational (line) and experimental (dots) [6] wave profiles alongside the hull for six different Froude numbers.

3. MESH STUDIES

The grid quality is fundamental for the convergence and accuracy of CFD calculations. Gird qualities are discussed in detail by Thompson et al[7] however, for this study the criteria available in ANSYS ICEM such as, *3x3x3 Determinant*, *Aspect Ration* and *Skewness*, were used to determine the mesh quality. Although it is quite difficult to set up all those criteria with ideal values, a right balance on them gives a good mesh quality. Generally speaking, a value of *3x3x3 Determinant* over 0.4, an *Aspect Ratio* between 100 and 500 and, a *Mesh Expansion Factor* with a maximum 50 are good indicatives that a mesh could work properly.

Having tested the conditions on the CFX solver, with result presented on the previous section, a new reference domain was made. The mesh strategy was to keep the same number of nodes (306,240) reducing the domain in the three Cartesian directions in order to obtain different levels of refinement, mainly close to the hull, Figure 4 show a sketch of the domains. Table 2 shows the dimensions of the domains in term of length and draught of the hull. In the longitudinal direction the measures show the length ahead and behind the hull (L=1m).

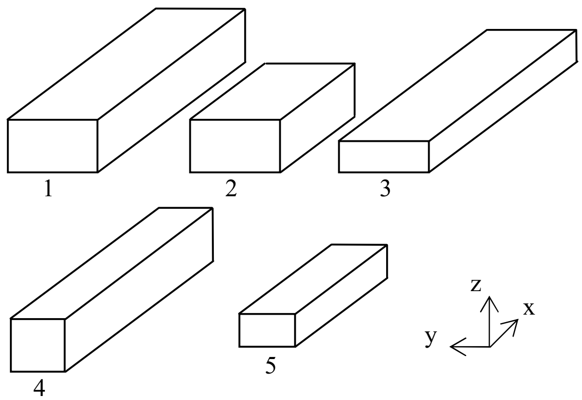

Figure 4. Domain sketches.

Table 2. Dimesion domains in terms of length and draught of the hull.

	Domain 1	Domain 2	Domain3	Domain 4	Domain 5
Length	L/2,L,1.2L	L/2,L,0.66L	L/2,L,1.2L	L/2,L,1.2L	L/2,L,0.66L
Width	L	L	L	L/2	L/2
Hight	L/2 + T	L/2 + T	L/3 + T	L/3 + T	L/3 + T

From the initial test, the highest percentage errors in total resistance coefficients were shown on the lowest and highest Froude numbers. Also, for those values, wave profiles along side the hull did not show good agreement with the experimental data, specially the height of the bow wave for the highest Froude number. Thus, Froude numbers of 0.250 and 0.408 were chosen to perform the mesh study for five domains and four different turbulence models. Figures 4 and 5 show the results, together with experimental data for those Froude numbers respectively.

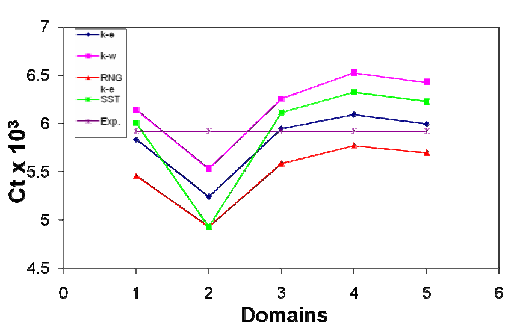

Figure 5. Total resistance coefficient for 5 domains and 4 turbulence models with 306,240 nodes each at Fn=0.250

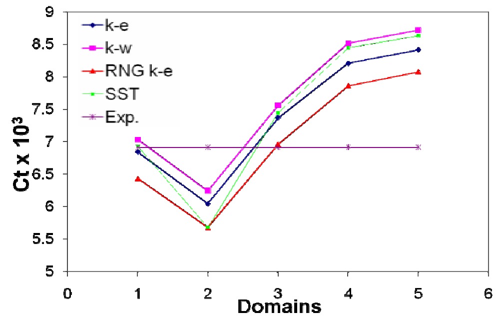

Figure 6. Total resistance coefficient for 5 domains and 4 turbulence models with 306,240 nodes each at Fn=0.40

4. VERIFICATION AND VALIDATION

As a preliminary step of verification, the longitudinal components of the normal and tangential forces alongside the hull can be used to give confidence on the value of the total drag, determined as the total force on the hull in the flow direction, see Table 1. However for specific Froude numbers the computational model could give misleading results. As a result it was decided to carry out mesh studies over different domains and different turbulence models at the lowest and highest Froude number, for this case 0.250 and 0.408 respectively.

From Figures 5 and 6, it could be said that domains 1 and 3 provide accurate results for resistance coefficients in both Froude numbers studied; this was also proved with percentage errors. The idea now is to determine which turbulence model fits better, taking the experimental data as an accurate resistance coefficient value. As it can be seen from those figures, the experimental data is a horizontal line with slope equals to zero and the intercept is the total resistance coefficient. In order to determine statistically the confidence of the conclusions for turbulence models, linear regressions were applied on the data for resistance coefficients and the following hypothesis tests were carried out. The domain 2 was discarded for the purpose of this analysis, as it was considered unusual since it produced a strange wave profile thus, just the other domains were taking into account for this analysis.

The hypothesis test[8] was applied regarding to the slope β_1 and to numerical data on linear regressions $y = \beta_1 x+\beta_0$, in this case experimental data is taken as a measure of validation pattern, hence

Ho: $\beta_1 = 0$ if $|t_o| < t_{\alpha/2, n-2}$ Numerical data slope is zero

Ha: $\beta_1 \neq 0$ if $|t_o| > t_{\alpha/2, n-2}$ Numerical data slope is not zero

Where:

- Ho: Null Hypothesis
- Ha: Alternative Hypothesis.
- $t_{\alpha/2, n-2}$ The value of the t-distribution for $\alpha/2$ with n-2 degree of freedom
- α : Significance level.
- t_o : The test statistic value calculated as shown:

$$t_o = \beta_1 \big/ \sqrt{M_{SE}/S_{xx}}$$

Where:

- M_{SE} : The mean square error
- S_{xx} : The sum of squares of the value minus the mean.

Ho: $\beta_o = C_t$ if $|t_o| < t_{\alpha/2, n-2}$ the numerical estimation is C_t

Ha: $\beta_o \neq C_t$ if $|t_o| > t_{\alpha/2, n-2}$ the numerical estimation is not C_t

t_o: The test statistic value calculated as shown

$$t_o = (\beta_o - C_t) \Big/ \sqrt{M_{SE}(1/n + (\bar{x})^2 / S_{xx})}$$

As a result of these tests, it could be said that turbulence models with less value of t_o produce closer values to the experimental results than those with high t_o values. Hence, for a Froude number of 0.250 the numerical total resistance coefficient is in accordance with the experimental data for the k-e model for the range of domains tested. Similarly, for the Froude number 0.408 the k-w model followed by SST predict betters the total drag.

5. CONCLUSIONS

The main aim of this study was to verify and validate the calculated hull resistance in terms of mesh and domain size, and to generate data to understand how many hull lengths are required ahead and behind the hull to obtain convergence and accurate data, which is not well defined by other authors. At least, for this case of a fixed Wigley hull, it has shown that even with short lengths between the stern and the outflow, accurate data can be obtained.

For the same number of elements, domains 1 and 3 showed better agreement with experimental data for total resistance coefficients. For the lowest Froude number, 0.250, the k-e turbulence model gave better agreement while for the highest Froude number, 0.408, the k-w model followed by SST predicted better the total resistance for those domains tested. However, the agreement for wave profiles at those Froude numbers tested did not improve.

FURTHER WORK

Simulations for more realistic hull forms under free to heave and pitch condition will be carried out based on the experience from the current work. Resistance data for verification and validation will be obtained from experimental work which will also measure the longitudinal wave cuts of the wave pattern instead of measuring the wave profile alongside the hull.

6. REFERENCES

[1] Zhang Z, Liu H, Zhu S, Zhao F. (2006) Application of CFD in ship engineering design practice and ship hydrodynamics. Conference of Global Chinese Scholars on Hydrodynamics.

[2] Gotman, A. (2007) Navigating the wake of past efforts. The Journal of Ocean Technology. Volume 2. Number 1. pp 74-96.

[3] The Resistance Committee (2005). Proceedings of the 24th IIT. Volume I. ITTC, 2005 U.K.

[4] Versteeg H.K, Malalasekera W. (1995). An introduction to Computational Fluid Dynamics. The finite Volume Method. Essex: Longman Scientific & Technical.

[5] CFX. CFX Manual V11. Ansys 2007.

[6] Kajitani, H et al (1983). The Summary of the Cooperative Experiments on Wigley Parabolic Model in Japan. Tokyo University. Japan

[7] Thompson J.F., Soni B.K., Weatherill N.P. (1999). Handbook of Grid Generation. CRC Press.

[8] Montgomery, D.(2005) Design and Analysis of Experiments. John Wiley & Sons, Inc.