
This version is available at https://strathprints.strath.ac.uk/63207/

Strathprints is designed to allow users to access the research output of the University of Strathclyde. Unless otherwise explicitly stated on the manuscript, Copyright © and Moral Rights for the papers on this site are retained by the individual authors and/or other copyright owners. Please check the manuscript for details of any other licences that may have been applied. You may not engage in further distribution of the material for any profitmaking activities or any commercial gain. You may freely distribute both the url (https://strathprints.strath.ac.uk/) and the content of this paper for research or private study, educational, or not-for-profit purposes without prior permission or charge.

Any correspondence concerning this service should be sent to the Strathprints administrator: strathprints@strath.ac.uk
Wave-Resistance Computation via CFD and IGA-BEM Solvers: A Comparative Study

Xinning Wang, Sotirios P. Chouliaras, Panagiotis D. Kaklis
Dept. Naval Architecture, Ocean and Marine Engineering (NAOME), University of Strathclyde
Glasgow, UK

Alexandros A.-I. Ginnis
Dept. Naval Architecture and Marine Engineering, National Technical University of Athens (NTUA)
Athens, Greece (GR)

Constantinos G. Politis
Dept. Naval Architecture, Technological Educational Institute of Athens (TEI-Athens)
Athens, GR

Konstantinos V. Kostas
Dept. Department of Mechanical Engineering, Nazarbayev University
Astana, Kazakhstan

ABSTRACT

This paper delivers a preliminary comparative study on the computation of wave resistance via a commercial CFD solver (STAR-CCM+®) versus an in-house developed IGA-BEM solver for a pair of hulls, namely the parabolic Wigley hull and the KRISO container ship (KCS). The CFD solver combines a VOF (Volume Of Fluid) free-surface modelling technique with alternative turbulence models, while the IGA-BEM solver adopts an inviscid flow model that combines the Boundary Element approach (BEM) with Isogeometric Analysis (IGA) using T-splines or NURBS. IGA is a novel and expanding concept, introduced by Hughes and his collaborators (Hughes et al, 2005), aiming to intrinsically integrate CAD with Analysis by communicating the CAD model of the geometry (the wetted ship hull in our case) to the solver without any approximation.

KEY WORDS: Computational Fluid Dynamics (CFD); Reynolds Averaged Navier Stokes (RANS) equations; Boundary Element Method (BEM), Isogeometric Analysis (IGA); wave resistance; Wigley ship; KRISO Container Ship (KCS).

INTRODUCTION

The prediction of wave resistance in naval architecture plays an important role in hull optimisation, especially for higher Froude numbers when wave-resistance’s share in total resistance becomes higher. It is well known that the total resistance of a ship can be roughly decomposed into the sum of frictional, viscous-pressure and wave resistance. Model testing is commonly used to predict the resistance components for new ships (ITTC, 1987). With the recent improvements in CFD (Computational Fluid Dynamics) tools, CFD is likely to provide a decent alternative for saving time and money for the prediction of resistance for modern ship hulls. This is not, however, the case for ship-hull optimisation when the geometry is unknown, which increases drastically the overall computational cost and the significance of deviation between the accurate CAD model of a ship hull and its discrete approximation usually adopted by the CFD solvers.

An alternative lower-cost path for the wave-resistance estimation can be employed by appealing to the Boundary Element Method (BEM) for solving the Boundary Integral Equation (BIE), which results from adopting the so-called Neumann-Kelvin model for the flow around an object moving on the otherwise undisturbed free-surface of an inviscid and irrotational liquid; see, e.g., (Brard, 1972) and (Baar and Price 1988). Our purpose is to initiate a systematic comparative study between a CFD solver (STAR-CCM+) and an in-house BEM solver enhanced with the IGA concept, which permits to tightly integrate the CAD model of a ship hull and its IGA-BEM solver; see, e.g. (Belibassakis et al 2013). Under the condition that this study will secure that the discrepancy between the results provided by the two solvers are acceptable within the operational range of Froude numbers, one can proceed to develop a hybrid mid-cost optimisation framework that combines appropriately the low-cost IGA-BEM solver (Kostas et al, 2015) with the high-cost CFD one. In the present paper our comparison will involve two hulls, namely the Wigley and the KCS hull, which have been extensively used in pertinent literature for experimental and computational purposes.

CFD SOLVER: METHODOLOGY AND SETUP

NAOME has provided (to the first three co-authors) access to the commercial CFD solver STAR-CCM+®, which uses a finite-volume method for capturing the free-surface elevation created by an object moving with constant velocity on the otherwise undisturbed free-surface of viscous incompressible fluid. This method uses a Volume of Fluid (VOF) approach based on integrating the incompressible Reynolds time–averaged Navier-Stokes (RANS) equations (Eq. 3, 4) over a control volume. Recall that the Navier-Stokes equations can be written as:
\[
\rho \left( \frac{\partial V_i}{\partial t} + (V \cdot \nabla) V \right) = -\nabla P + \mu \nabla^2 V + f
\]  
(1)

\[
\frac{\partial V_i}{\partial t} + \frac{\partial}{\partial x_j} (V U_j) = - \frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( 2 \nu S_{ij} - u_i u_j \right),
\]
(2)

where \(V = (V_x, V_y, V_z)\) is the fluid-velocity vector, \(\rho\) is the fluid density, \(\mu\) is the dynamic viscosity and \(f\) represents the external forces acting on the fluid. The associated RANS equations can then be written in tensor form as:

\[
\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_j} (U_j U_i) = - \frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( 2 \nu S_{ij} - u_i u_j \right),
\]
(3)

\[
\frac{\partial U_i}{\partial x_i} = 0,
\]
(4)

where \(U_i\) stands for the mean flow velocity component \((i=1,2,3)\), \(v\) is the kinematic viscosity, \(S_{ij}\) is the mean strain-rate tensor given by:

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)
\]
(5)

and, finally, \(u_i u_j\) is the Reynolds stress tensor \(R_{ij}\). The well-known closure problem of RANS equations consists in modelling the Reynolds stress tensor as a function of the mean velocity and pressure, in order to remove any reference to the fluctuating part of the velocity. In this work, we employ two of the most common turbulence models used in CFD, namely the k-epsilon \((k-\varepsilon)\) model and the k-omega \((k-\omega)\) model. The k-epsilon model is a two equation model which gives a general description of turbulence by means of two transport partial differential equations; see, e.g., (Lauder and Spalding 1974). The k-omega model attempts to predict turbulence by two partial differential equations in terms of two variables, namely \(k\) and \(\omega\), with \(k\) being the turbulence kinetic energy while \(\omega\) is the specific rate of dissipation of the turbulence kinetic energy \(k\) into internal thermal energy; see, e.g., (Wilcox 2008).

Locating the free surface in the two-phase (air, liquid) flow, created by the movement of a body on the free-surface of a fluid, can be materialised via the so-called Volume Fraction Transport equation (Peric & Ferziger, 2002) given below

\[
\frac{\partial c}{\partial t} + \frac{\partial (c U_j)}{\partial x_j} = 0
\]
(6)

where the volume fraction \(c\) is equal to \(V_{\text{air}}/V_{\text{total}}\) and the fluid density \(\rho\) is equal to

\[
\rho = \rho_{\text{air}} c + \rho_{\text{water}} (1-c),
\]
(7)

\[
\mu = \mu_{\text{air}} c + \mu_{\text{water}} (1-c).
\]
(8)

According to the standard practice, the total resistance of a ship is subdivided into two components, namely:

\[
C_T = C_F + C_R,
\]
(9)

where \(C_T\) is the total resistance coefficient, \(C_F\) is the friction resistance coefficient and \(C_R\) is the residual resistance coefficient. The friction resistance coefficient depends only on Reynolds number \(Re\) and assumed to be independent from the residual resistance coefficient. Residual resistance (coefficient) can be further decomposed into wave resistance \(C_W\) and viscous pressure resistance \(C_{VP}\) coefficients, resulting in:

\[
C_T = C_F + C_{VP} + C_W.
\]
(10)

In the context of the the resistance test method adopted by the International Towing Tank Conference (ITTC) on 1978, the concept of form-factor \(k\) has been introduced, based on two assumptions, i.e., invariance between the model and the full-scale ship and invariance with respect to the Froude number \(Fr\). Working in this context, we can write

\[
C_{VP} = k C_F,
\]
(11)

which results in:

\[
C_T = (1 + k) C_F + C_W.
\]
(12)

For the two case studies undertaken in this paper, the form-factor for the Wigley hull will rely on experimental values from (Ju 1983) while the form-factor for the KCS hulls will be based on experimental results and CFD estimates.

Wigley hull is a biquadratic surface expressed analytically as:

\[
y(x, z) = \frac{6}{7} \left( 1 - \left( \frac{2x}{L} \right)^2 \right) \left( 1 - \left( \frac{z}{T} \right)^2 \right),
\]
(13)

where \(L=4.0m\) (length between perpendiculars), \(B=0.4m\) (breadth), \(T=0.25m\) (draft), while \(x\) identifies the distance from mid-ship (positive towards bow), \(y\) denotes the distance from the symmetry plane and \(z\) denotes the distance measured from the undisturbed free-surface.

The second case study is a model of the so-called KCS (KRISO container ship) with main particulars given in Table 1. The CFD solver is using the NURBS-based CAD model (see Fig. 1) of the KCS ship which is available at the web-site of NMRI (National, Maritime Research Institute) of Japan1. For the needs of the IGA-BEM solver a new CAD model (see Fig. 2), as a multi patch NURBS model of the KCS model has been rebuilt for the hull below the waterline. This CAD model comprises bi-cubic patches and possesses first-order \((G^1)\) geometric continuity globally, i.e., continuously varying unit normal. The surface is generated with a lofting (skinning) scheme on mid-body sections where the remaining stern/bow patches are the result of Gordon surface constructions on the corresponding sections, waterlines and/or stern/bow profile parts. The deviation between the two CAD models below the design waterline, measured in terms of integral geometric characteristics, is indeed very small, i.e., wetted-surface deviation: 0.076%, wetted-surface deviation: 0.055%, centroid deviation: (-0.010, 0.000, -0.037)%.

<table>
<thead>
<tr>
<th>Table 1. Main particulars of the KCS ship</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model</td>
</tr>
<tr>
<td>KCS ship</td>
</tr>
</tbody>
</table>

1 https://www.nmri.go.jp/institutes/ship_performance_evaluation/cfd_rd/
Meshing in the 2-phase flow region is undertaken by the adopted CFD solver enabling us to create trimmed hexahedral grids and prism layers along walls. Trimmed grids allow anisotropic local refinement around the hull and the free-surface. Representative 2D intersections of the developed meshes with appropriate planes are given in Figs. 3 to 6 while Tables 2 and 3 provide mesh-size information.

Table 2. Fine-mesh information of the Wigley ship

<table>
<thead>
<tr>
<th>min. element size</th>
<th>0.06 (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>max element size</td>
<td>0.48 (m)</td>
</tr>
<tr>
<td># elements</td>
<td>1,648,435</td>
</tr>
</tbody>
</table>

Figure 3: Top-view of the mesh around the Wigley hull, showing different levels of refinement in the Kelvin-angle cone.

Figure 4: Transverse intersection of the mesh around the midship section of the Wigley hull, showing local refinement near the free-surface.

Taking into account the symmetry of the flow with respect to the centre plane, the axes-aligned bounded boxes, used for by the CFD solver as computational domain, is the box [-5L,2.5L]x[0L,3.75L]x[-3.75L,2L] for the Wigley hull and the box [-2.47L,2.47L]x[0L,2.47L]x[-2.47L,1.24L] for the KCS ship hull, with L denoting the length of the corresponding hull. On the boundary of these computational boxes the typical boundary conditions in CFD problems are imposed, such as, inlet/outlet, wall, constant-pressure, symmetry boundary conditions, etc.

In order to choose the appropriate element base size and turbulence model, CFD results will be compared against available experimental results for Froude number $Fr=0.267$ for the Wigley hull and $Fr=0.26$ for both the original and the rebuilt KCS hulls. For this purpose, the CFD tool is used to compute the total force acting on the hull in the direction of its motion ($x$-direction) and then non-dimensionalised using the below formula, where $A_W$ is the static wetted surface area of the hull and $U_0$ is the tow velocity.

$$C_t = \frac{R_F}{0.5 \rho A_W U_0^2}$$

Experimental results for the Wigley hull are available in (Ju 1983), where $U_0=1.67$ m/s, $A_W=Cs\cdot L(2D+B)$, $Cs=0.661$, and $C_t=4.16\times10^{-3}$ where $C_t$ denotes the total-resistance coefficient. For the KCS hull: $A_W=9.4379$ m$^2$ and $U_0=2.196$ m/s. Appealing to (Kim et al 2001), we have that $C_t=3.557\times10^{-3}$ while the frictional-resistance coefficient, $C_F$, is calculated using the ITTC correlation line (ITTC, 2008a) resulting in $C_F=2.832\times10^{-3}$. The following three tables summarise a grid sensitivity analysis of the three test hulls with respect to the base size of the mesh adopted by the CFD tool and the employed turbulence models. Refinement is based on the pattern $coarse\_size=\sqrt{2}$, $medium\_size=2$, $fine\_size$ as recommended by ITTC (2008b). Tables 4 and 5 indicate that, for the k-e turbulence model, percentage error decreases as we move from coarse to fine mesh, which is however achieved via a dramatic increase in time cost. For the KCS test case the significant decrease of percentage error should be attributed to the “alignment” of fine mesh with the “needs” of the turbulence model. On the other hand, the error seems to be mesh-invariant for the turbulence model of Table 6.
Table 4. Grid sensitivity analysis for the Wigley hull \((Fr=0.267, k-e\) turbulence model)

<table>
<thead>
<tr>
<th>grid</th>
<th>base size (m)</th>
<th>#cells (M)</th>
<th>(C_T \cdot 10^3) (error%)</th>
<th>time (h/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>coarse</td>
<td>0.1200</td>
<td>0.48</td>
<td>3.83 (7.87%)</td>
<td>3/15</td>
</tr>
<tr>
<td>medium</td>
<td>0.0850</td>
<td>0.77</td>
<td>3.90 (6.25%)</td>
<td>6/12</td>
</tr>
<tr>
<td>fine</td>
<td>0.0600</td>
<td>1.6</td>
<td>4.35 (4.46%)</td>
<td>10</td>
</tr>
</tbody>
</table>

Experimental value: \(C_T = 4.16 \times 10^{-3}\), \#cores = 12

Table 5. Grid sensitivity analysis for the KCS hull \((Fr=0.26, k-e\) turbulence model)

<table>
<thead>
<tr>
<th>grid</th>
<th>base size (m)</th>
<th>#cells (M)</th>
<th>(C_T \cdot 10^3) (error%)</th>
<th>time (h/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>coarse</td>
<td>0.1125</td>
<td>0.86</td>
<td>3.85 (8.23%)</td>
<td>4/8</td>
</tr>
<tr>
<td>medium</td>
<td>0.0800</td>
<td>1.7</td>
<td>3.78 (6.15%)</td>
<td>10/11</td>
</tr>
<tr>
<td>fine</td>
<td>0.0560</td>
<td>2.1</td>
<td>3.54 (-0.51%)</td>
<td>16/30</td>
</tr>
</tbody>
</table>

Experimental value: \(C_T = 3.557 \times 10^{-3}\), \#cores = 12

Table 6. Grid sensitivity analysis for the KCS hull \((Fr=0.26, \text{Shear Stress Transport (SST) eddy viscosity model blending a variant of the } k-\omega \text{ model in the inner boundary layer and a transformed version of the } k-e \text{ model in the outer boundary layer and the free stream})\)

<table>
<thead>
<tr>
<th>grid</th>
<th>base size (m)</th>
<th>#cells (M)</th>
<th>(C_T \cdot 10^3) (error%)</th>
<th>time (h/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>coarse</td>
<td>0.1125</td>
<td>0.86</td>
<td>3.79 (6.68%)</td>
<td>5/10</td>
</tr>
<tr>
<td>medium</td>
<td>0.0800</td>
<td>1.7</td>
<td>3.74 (5.03%)</td>
<td>8</td>
</tr>
<tr>
<td>fine</td>
<td>0.0560</td>
<td>2.1</td>
<td>3.80 (6.70%)</td>
<td>16</td>
</tr>
</tbody>
</table>

Experimental value: \(C_T = 3.557 \times 10^{-3}\), \#cores = 12

Finally, the ensuing three figures illustrate the performance of the CFD solver for estimating the wave/total and residual resistance of the Wigley and KCS hull against experimental results provided in (Ju 1983) and (Choi et al 2011), respectively. The wave resistance estimate in Fig. 8 is obtained by subtracting from the computed total-resistance coefficient the viscous resistance approximated by \((1+k)C_F\), where \(C_F\) is the well-known ITTC-57 friction-resistance coefficient and \(k=0.08\), which is obtained by applying Prohaska’s method in conjunction with CFD total resistance estimates for small Froude numbers. Note that \(k=0.1\) according to an experimental study available in (Ju 1983). As for Fig. 10, the CFD estimate of the residual resistance is obtained by subtracting from the total-resistance coefficient \(C_T\) the frictional-resistance coefficient \(C_r\), evaluated again via the ITTC-57 formula.
IGA-BEM WAVE-RESISTANCE SOLVER

This paper follows the approach by Belibassakis et al (2013) based on the formulation by Brard (1972) and Baar and Price (1988). The ship-hull sails through an incompressible and inviscid fluid with a uniform velocity formulation by Brard (1972) and Baar and Price (1988). The total velocity of the flow consists of the uniform velocity and the perturbation velocity due to the existence of the hull. The problem can be formulated by the weakly singular BIE, 

\[
\frac{\mu(P)}{2} = \frac{1}{k} \int_{\partial S} \frac{\partial G(P,Q)}{\partial n(Q)} n(Q) \, dS(Q) \quad \text{for} \quad P, Q \in S
\]

where \(\mu\) is the density of the source distribution on the hull surface, \(G\) is the Neumann-Kelvin Green’s function corresponding to a point source moving with velocity \(U\) on the undisturbed free surface. Furthermore, \(G^*\) is the regular part of Green’s function, \(k\) is the characteristic wave number, \(n\) is a vector normal to the boundary surface \(S\) on point \(P\), \(t\) corresponds to the waterline and \(n_t\) and \(t_t\) are the vectors normal and tangent to the waterline respectively on point \(Q\).

\[
P, Q \in S
\]

The discrete form of BIE (Eq. 15) is 

\[
\mu(t_1, t_2) = \sum_{i=0}^{n_1} \sum_{j=0}^{n_2} \mu_i \phi_j R_{ij}^P(t_1, t_2), \quad \text{for} \quad i, j \in I_1 \times I_2,
\]

where \(\mu_i\) are the unknown source density coefficients and \(\Pi = (I_1^p, I_2^p)\) where \(I_1^p, I_2^p\) are the numbers of added knots for each parametric direction. The accuracy of this method depends on the number of the source density coefficients which are essentially the degrees of freedom (DoFs) of the numerical procedure. Consequently, a number of DoFs may be added by knot insertion given by \(P^p\) in order to get a more accurate approximation of the solution. The total number of DoFs is given by 

\[
M = \sum_p \left( n_1^p + I_1^p + 1 \right) \left( n_2^p + I_2^p + 1 \right)
\]

Eq. 15 will be numerically solved by applying a collocation point scheme where each collocation point \(P^p\) on the physical space corresponds to the so-called Greville abscissae Farin (1999) of the associated knot vectors.

For each collocation point \(P^p\), the induced velocity factor can be evaluated by 

\[
u_i(P^p) = \int_{t=1} \left[ \nabla_{t} G(P^p, x(t)) \right] \frac{\partial \alpha(t, t_1)}{\partial t_1} \, dt_1, \quad p, q = 1, 2, \ldots, N,
\]

where \(\alpha\) is the determinant of the surface metric tensor. These factors represent the velocity at \(P^p\) induced by a source distribution of density \(\Phi_i^p\). Evaluation of Eq. 19 can be achieved by assuming that each induced factor is introduced as:

\[
u_i(P^p) = \nu_i^{(\text{non-sing})}(P^p) + \nu_i^{(\text{sing})}(P^p)
\]

The non-singular integrals are easily calculated by using standard quadrature methods. On the other hand, singular integrals are divided into three cases:

- **Far field**: Quadrature methods are used as in the non-singular case.
- **Near-field**: Transformation techniques are used as in Telles (1987) and (1994)
- **In-field**: Cauchy – Principal Value integrals occur and are evaluated as in Mikhlin (1965).

The integration of the non-singular parts when points \(P\) and \(Q\) approach the free-surface \(z=0\) becomes numerically unstable, as observed by Belibassakis et al (2013). As a result, the ship is considered to be furtherly submerged by \(z=0\) can be evaluated by numerical experimentation and its value varies for different hull shapes.

The discrete form of BIE (Eq. 15) is 

\[
\sum_{i=0}^{n_i} \mu_i^p R_{ij}^q \left( t_1^p, t_2^p \right) - 2\Pi(P^p) = -2U(P^p) \cdot n(P^p), \quad j = 0, \ldots, n_q + 1', \quad p = 1, \ldots, N
\]

where 

\[
\Pi(P^p) = \sum_{q=0}^{N-1} \sum_{i=0}^{n_i} \mu_i^p \left[ u_i^q(P^p) \cdot n(P^p) \right]
\]

where \(u_i^q\) are the induced velocity factors and \(n\) is the unit vector normal to the boundary surface. Convergence usually requires more degrees of freedom than those offered by the representation of the geometric model and thus, the NURBs bases may need refinement. In the context of this
work, knot insertion has been applied (h-refinement) but degree elevation (p-refinement) or both knot insertion and degree elevation (k-refinement) may be utilised.

After the linear system is solved, the velocity of each collocation point can be evaluated by

$$\mathbf{v}(P_i) = \mathbf{U} + \mathbf{u}(P_i)$$  \hspace{1cm} (23)

where $\mathbf{U}$ is the ship’s speed and $\mathbf{u}$ is the induced velocity of collocation point $P_i$ given by

$$\mathbf{u}(P_i) = \sum_{q=1}^{N} \sum_{k=0}^{N} \mu_k^q \left( \frac{1}{2} R_{i,j}^q \left( t_{i,j}^q + t_{i,j}^q \right) \right) \mathbf{n}(P_i) + \mathbf{u}_0(P_i)$$

$$j = 0, \ldots, n + 1, \quad p = 1, \ldots, N$$  \hspace{1cm} (24)

And finally the wave resistance can be calculated by

$$C_W = \frac{R_w}{0.5 \rho U^2 S} = \frac{1}{N} \left( \sum_{i=1}^{N} C_p n_i \sqrt{a} dt_i dt_2 \right)$$  \hspace{1cm} (25)

where $S_w$ is the wetted surface of the hull, $n_i$ is the x-component of the vector normal to the boundary surface $S$ and $C_p$ is the pressure coefficient given by

$$C_p = \frac{P - P_0}{0.5 \rho U^2} = 1 - \left( \frac{V}{U} \right)^2 - 2gz / U^2,$$  \hspace{1cm} (26)

This method has been implemented for the Wigley and the rebuilt KCS hulls, presented in the previous section. An example of the pressure coefficient distribution on the Wigley hull for $F_n=0.316$ can be found in the below figure. In this connection it should be stressed that $C_p$ is evaluated directly on the ship hull via the same set of bases functions used for building the CAD model of the hull.

Fig. 12: Pressure coefficient distribution $C_p$ over the Wigley hull for $F_n=0.316$

COMPARISON & CONCLUSIONS

Figures 9 and 10 collect CFD and IGA-BEM estimates of the wave-resistance coefficient for the Wigley hull and the residual-resistance coefficient for the KCS hull. In the latter case, the IGA-BEM estimate for residual resistance is obtained by adding to the wave-resistance coefficient a viscous pressure correction term of the form $kC_F$, where $C_F$ is the ITTC correlation line (ITTC, 2008a) and $k$ is a form factor that, if not available from experimental data, can be estimated with the support of Prohaska’s method and the CFD tool. One can generally assert that the IGA-BEM resistance curves are shape aligned with both the corresponding CFD curves and the experimental data with an average error of 6.4% for the wave resistance estimate over the Froude numbers for which experimental results are available. In this connection, the results of the present preliminary study towards the feasibility of developing a hybrid mid-cost optimisation framework that combines a low-cost IGA-BEM solver with the high-cost CFD one, are not discouraging.

Fig. 13: Simulation & experimental results for the Wigley ship

Fig. 14: Simulation and experimental results for the KCS ship

ACKNOWLEDGEMENTS

CFD Results were obtained using the EPSRC funded ARCHIE-WeSt High Performance Computer (www.archie-west.ac.uk). EPSRC grant no. EP/K000586/1. S.P. Chouliaras and P.D. Kaklis received funding from the European Union’s Horizon 2020 research and innovation programme under the Marie Skłodowska-Curie grant agreement No 675789
REFERENCES


Brard, R (1972). The representation of a given ship form by singularity distributions when the boundary condition on the free surface is linearized, J. Ship Res. 16, 79-82.


