XI International Conference on Computational Methods in Marine Engineering

Scale effect analysis on calm water resistance using model basin testing conditions and CFD

S. Cunningham[1*] and J. Green[1]

[1]BMT, 1 Park Road, Teddington, London, TW11 0AP, UK

[*] Corresponding Author: shona.cunningham@uk.bmt.org

ABSTRACT

The work analyses the flow around a complex hull form and demonstrates the viability of using Computational Fluid Dynamics (CFD) for full scale resistance calculations during ship design. The analysis examines the results of physical model testing at six different scales at a constant Froude number of 0.219 in comparison to a CFD derived solution. The multi-purpose CFD software Ansys CFX is used for the analysis, employing the standard non-homogenous Volume of Fluid (VOF) multiphase method. As the physical model testing did not consider sink and trim, the CFD solution did not consider sink and trim. The analysis validates the use of model scale CFD through comparing the total drag resistance to model basin tests, with the results from the two approaches within 3.95% of each other at all six scales as shown in Figure 1. Using the International Towing Tank Conference (ITTC) scaling method, the coefficient of full scale total resistance is calculated using the ITTC 78 friction line and a CFD friction line, and these values are compared with one another. The paper discusses the uncertainties associated with the scaling methodologies in comparison to direct full scale CFD calculations and the effect of the frictional line on the total coefficient of resistance obtained. Also highlighted is the importance of the Reynolds number in the scaling process.

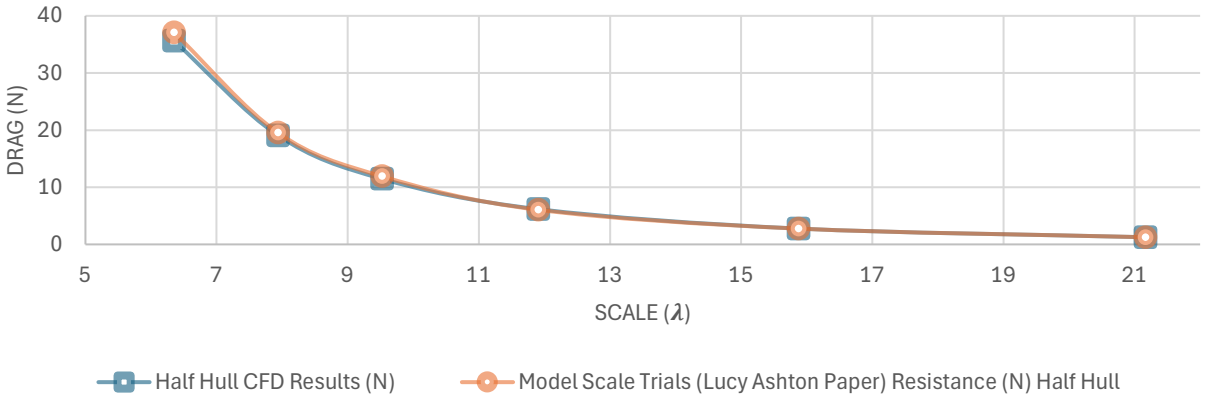

Figure 1: Comparison of half of the total resistance recorded for a symmetric hull from the model basin test and the CFD model scale analysis