A CASE STUDY ON STABILISER FIN-HULL INTERACTION USING CFD AND MODEL EXPERIMENTS

(DOI No: 10.3940/rina.ijme.2015.a4.333)

K A Bhosale and **J T Duffy**, Australian Maritime College, Australia

SUMMARY

A case study was conducted to investigate and quantify stabiliser fin-hull interaction using a combination of Computational Fluid Dynamics and physical model experiments. The fin-hull interaction was studied by comparing the lift and drag of a stabiliser fin in a free stream condition and when attached to a hull. The findings of this case study showed that using free stream fin characteristics to predict performance of a stabiliser fin fitted to the hull resulted in an over-prediction of drag by up to 46% and under-prediction of lift by up to 75% for the speeds and angle of attack analysed. These discrepancies are for this case study only and in practice will vary for different hull forms, fin types, fin location and angles of attack. However, the research highlights the limitations of using free stream fin characteristics to predict the performance of a fin fitted to a hull.

NOMENCLATURE

AoA	Stabiliser fin angle of attack (deg)
AR	Geometric aspect ratio
C_D	Fin drag coefficient ($D / 0.5\rho SV^2$)
C_L	Fin lift coefficient ($L / 0.5\rho SV^2$)
D	Fin drag (N)
D''_{Hull}	Non-dimensional hull drag (N)
Fr	Froude number
g	Gravity constant (9.81 m/s^2)
L	Fin lift (N)
L''_{Hull}	Non-dimensional hull lift (N)
L_{WL}	Hull waterline length (m)
Re	Reynolds number
S	Stabiliser fin projected (geometric) area (m^2)
V	Hull speed, same as free stream velocity (m/s)
∇_{Hull}	Hull volumetric displacement (m^3)
λ	Model hull scale ratio
ρ	Water density (kg/m^3)

1. INTRODUCTION

Fin stabilisers are commonly used to reduce vessel motions in a seaway. They work by generating dynamic lift which opposes the heave force exerted on the vessel in a seaway. When located away from the centre of gravity the fins also provide moments to oppose the vessels pitch and roll moments. During preliminary ship design, stabiliser fins are often sized by calculating the wave forces and moments acting on the vessel based on sea state and vessel hydrostatics. The fins are then designed to generate a lift force that is sufficient to counter the wave induced forces and moments. The common design method described in text books [1], is to predict the hydrodynamic lift generated by the fin using lift coefficient curves produced from free stream experiments in wind or water tunnels. One limitation with this approach is that a fin in free stream behaves quite differently to a fin fitted to a hull which is influenced by factors such as hydrodynamic pressure distribution and the magnitude and direction of flow velocity around the hull. These effects have been collectively termed as fin-hull interaction in the present study. Therefore, using free stream data to predict the lift and drag of a fin attached to a hull may result in the actual fin performance differing from expectations. In preliminary and final ship design, understanding fin-hull interaction is deemed important in order accurately to predict fin performance and hence design an efficient stabiliser. The aim of this research was to study and quantify the fin-hull interaction for the case of fin stabilisers for a trawler hull.

A significant amount of literature and experimental data, such as lift and drag coefficient (C_L and C_D) curves, is available for common foil sections including the NACA 0015 used in this research [2,3]. However, for low aspect ratio (AR) fins operating at low Reynolds number (Re) such as ship rudders and stabilisers, one of the most recognized studies is the set of wind tunnel experiments conducted by Whicker and Fehlner, who tested several low AR fin shapes and cross sections and presented C_L and C_D curves for a range of Re [4]. This is widely regarded as one of the most comprehensive set of free stream results for low AR fins and was used in this research to validate fin performance in free stream conditions. Literature specific to experiments involving fin stabilisers attached to a model ship hull is relatively scarce. The most recent study was conducted by Gaillarde who tested three different free running model hulls with stabilisers in a model test basin to determine the fin drag [5]. Similarly, the drag of stabilisers when attached to a hull was also determined by Della Rosa using Computational Fluid Dynamics (CFD) [6]. Della Rosa used Reynolds Averaged Navier Stokes (RANS) equations to determine stabiliser fin drag in steady flow and found the CFD predictions showed good agreement with physical model test results which verified the use of RANS method for such analysis. However, based on a review of published literature and the authors' best knowledge, studies to determine and quantify fin-hull interaction are either very scarce or have not been undertaken before. This is most likely because such studies are closely linked with relatively difficult research aspects such as modelling and prediction of hull boundary layer and turbulence near the stabiliser. As such, this research was conducted with an aim to analyse fin-hull interaction through a case study, using a combination of CFD and physical model experiments. The findings of this case study will hopefully contribute to better understanding and quantification of fin-hull interaction when analysing lift and drag of stabiliser fins fitted to a hull.

2. METHODOLOGY

2.1 RESEARCH METHODOLOGY

The approach used to achieve the aims of this research is shown in Figure 1. A combination of empirical, numerical and experimental methods was used to determine and validate fin lift and drag in different conditions. The fin was first analysed in free stream condition using a CFD model that was validated against the wind tunnel experiments results from [4]. Physical model experiments were then conducted in the Australian Maritime College (AMC) towing tank (TT) on a bare hull model and on the same hull with two fins attached (port and stbd). Finally, CFD analysis was undertaken to simulate the conditions with model fins on the hull as tested during TT experiments. These results were validated against the TT experiments results and the validated CFD model was used to determine the full scale performance of the fin when fitted to the hull. The performance of the full scale fin in free stream and when attached to the hull was then compared to quantify fin-hull interaction. In conjunction with the TT experiments, the CFD simulations allowed better flow visualization and understanding of the flow characteristics around the hull and in particular at the stabiliser fin.

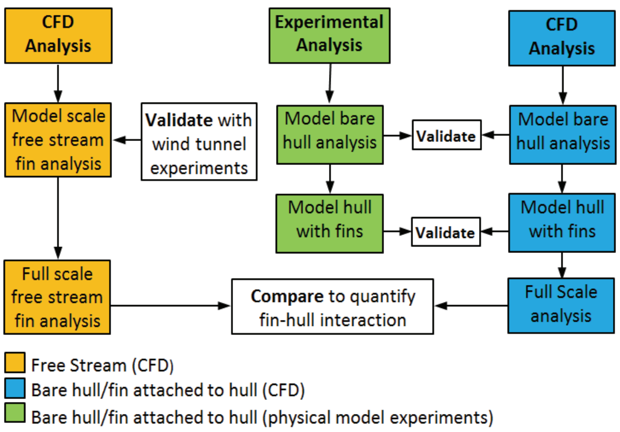


Figure 1: Approach used to quantify fin-hull interaction.

2.2 TERMINOLOGY AND PARAMETERS

In the context of this case study and the findings presented in this paper, it is important to clearly outline the terminology and key parameters used in the analysis. Fin-hull interaction refers to the change in fin lift and drag due to the fin operating on the hull. This interaction includes (but is not limited to) influence from factors such as the hydrodynamic pressure distribution, the magnitude and direction of flow velocity around the hull and fin submergence. Further breakdown and analysis of the individual factors contributing to fin-hull interaction was outside the scope of this research. The performance of the fin was determined as lift (L) and drag (D) force and analysed as non-dimensional coefficients C_L and C_D, where $C_L = L / 0.5\rho SV^2$, $C_D = D / 0.5\rho SV^2$, S = fin projected area (m^2), V = hull velocity (m/s) and $\rho = \rho_{FW}$ for model scale fins and ρ_{SW} for full scale CFD simulations. The derivation of fin L and D force in CFD simulations and physical model experiments is described later in the paper. The sign convention for hull and fin L and D used in this paper is presented in Figure 2. In line with the research aims, it was also necessary to compare the performance of the fin in free stream and when attached to the hull at both model and full scale. Since Froude scaling is typically used in model testing of surface craft, all fin performance was compared as a function of the hull Fr. For consistency and clarity, the term 'Free Stream Fr' was used, such that $Fr_{\text{Free Stream}} = Fr_{\text{Model Hull}} = Fr_{\text{Full Scale Hull}} = V / (g\, L_{WL})^{0.5}$ where L_{WL} = hull waterline length (m).

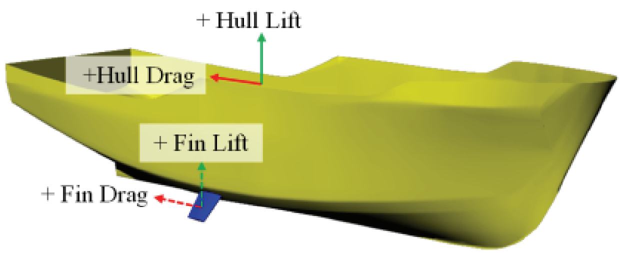


Figure 2: Fin and hull lift and drag sign convention used in this research.

2.3 SIMPLIFICATIONS AND LIMITATIONS

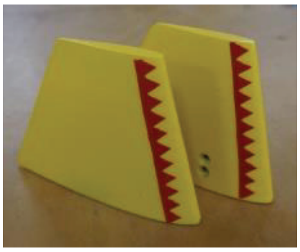
Analysis of active stabilisers in unsteady flow involves complex non-linear and unsteady (transient) variables. This was outside the scope of this research. Hence, this case study was conducted in calm water, with stabilisers set to 15° angle of attack (AoA) and the model hull constrained in 6 degrees of freedom (DOF) at the design waterline. The constraints ensured the hull sinkage and running trim were constant (zero) for both bare hull and finned hull. This also correlated with the simplifications made during the CFD analysis. The 15° AoA was chosen based on separate CFD analysis conducted to determine the optimum AoA for maximum lift and drag without the risk of dynamic stall. This was predominantly for the TT experiments where the physical measurement of small forces is limited by accuracy and tolerances of the load cells used to measure the forces. For the same reason, TT experiments and subsequent analysis was limited to Fr = 0.118 to 0.412 (ship speed of 4-14kts). Note that this is an idealized condition used in this particular case study and was used in order to analyse the fin performance in a controlled environment to isolate the influence of specific variables. In reality, the fin performance will also be influenced by factors such as vessel motion when underway in a seaway and varying angles of attack for active fin stabiliser systems. Considering these simplifications and limitations, it is necessary to highlight that the results and discussion presented in this paper are based on the findings of this case study alone. Where appropriate, the relevance of these findings to wider research and industry applications has been discussed. However, further work is necessary before applying or drawing generalized conclusions based on the information presented in this paper.

3. PHYSICAL MODEL EXPERIMENTS

3.1 SHIP AND FIN MODELS

A 1:20 scale geosim of FTV Bluefin, a fishing trawler owned and operated by the AMC, was used in the TT experiments (model AMC 96-01). The model had a L_{WL} of 1.57m, beam of 0.5m, draft of 0.2m and was chosen for its relatively small scale ratio (λ) which made it possible physically to test the 1:20 scale fins in the AMC TT. The model was prepared in accordance with ITTC procedure 7.5-01-01-01 [7] and the hull was fitted with turbulence stimulators (3mm×3mm round brass studs) at $0.05L_{WL}$ aft of the forward perpendicular. To ensure validity of the research, stabiliser fins were specifically designed for Bluefin using a design iterative method [1]. Based on the vessel hydrostatics data [8] and full scale seakeeping trials conducted onboard Bluefin [9], a required full scale fin projected area (S) of 2.72 m^2 was calculated. The fin was then designed with a NACA 0015 cross section, a geometric AR of 1, full scale mean chord of 1.65m and a taper ratio of 0.45. This was an exact geosim of the fin tested by Whicker and Fehlner in the wind tunnel experiments [4] and allowed full 3D validation of free stream CFD predictions. The 1:20 scale model fin used in the TT experiments was manufactured (3D printed) out of ABS M30 and mounted on the model hull using custom mounting brackets and aluminium shafts to secure the fin at 15° AoA as shown in Figure 3a.

a) Fin fitted to the hull at 15° AoA



b) Hama strip turbulence stimulators on fins

Figure 3: Fins attached to the model hull and fins with Hama strip turbulence stimulators

Hama strip turbulence stimulators were fitted to the fins at 5% of local chord length behind the leading edge in accordance with ITTC recommendation [10] as can be seen in Figure 3b. The geometry and location of the strips was based on available experimental data for low AR NACA 0015 fins [11] and the critical strip height was determined using the empirical formulae based on local fin Re [12].

3.2 WORK PROGRAM

Three different conditions were tested in the AMC TT as shown in Table 1. The tests were performed in calm water with the fin set to 15° AoA as described earlier. A standard bare hull calm water resistance (CWR) test was initially conducted (Condition 1) as a bench mark. With the model free to heave and trim, the hull drag, sinkage, and running trim was determined and validated against known CWR test results for the AMC 96-01 model hull. This confirmed equipment calibration and setup of the TT carriage. The remaining two conditions were then tested with the same equipment and carriage setup. The standard CWR test method stipulated in ITTC procedure 7.5-02-02-01 [13] was used for all tests. The only exception was that the model was constrained in 6DOF for Conditions 2 and 3. The net vertical (lift) and horizontal (drag) forces acting on the model hull were measured using two multi axis load cells mounted between the hull and towing posts. The fin lift and drag was determined as the difference between the vertical and horizontal forces measured for the baseline bare hull condition (Condition 2) and hull with fins (Condition 3). An experimental uncertainty analysis was performed in accordance with ITTC procedure 7.5-02-02-02 [14]. For the fin, the total uncertainty for fin C_L and C_D was taken as a function of the fin lift and drag force, fin projected area and model hull velocity. As such, the total uncertainty varied for each run (at different speeds) and for each condition tested in the TT.

Table 1 - Physical model experiment test program

Condition Number	Fins fitted	Constrained in 6DOF
Condition 1	No	No
Condition 2	No	Yes
Condition 3	Yes	Yes

4. CFD SIMULATIONS

The CFD analysis was conducted in two separate parts, as illustrated in Figure 1. This included analysing the performance of the model and full scale fin in a free stream condition and when fitted to the hull which was modelled to include the free surface. All domains were discretized using a tetrahedral unstructured mesh and solved in ANSYS CFX v.15 using the RANS method and the RANS Shear Stress Transform turbulence model. The CFD numerical computations were performed using an 8 core 16GB 3.4GHz computer which was deemed sufficient for all simulations. The free stream conditions were modelled with the fin attached to a free slip wall (hence no boundary layer at fin root) and the domain boundaries set to have no influence on fin performance. The free stream model and mesh are shown in Figure 4. A body of influence (BOI) with a high density mesh was modelled at the fin to capture the local pressures and velocities around the fin. Richardson Extrapolation method was used to determine the mesh uncertainty and to select the optimum mesh size. Based on this, a 0.8×10^6 element mesh was chosen which gave a maximum mesh uncertainty of 10% for lift and 2% for drag. The CFD simulation with the fins on hull was set up to replicate the test conditions during the TT experiments. This included bare hull (Condition 2) and hull with fins (Condition 3).

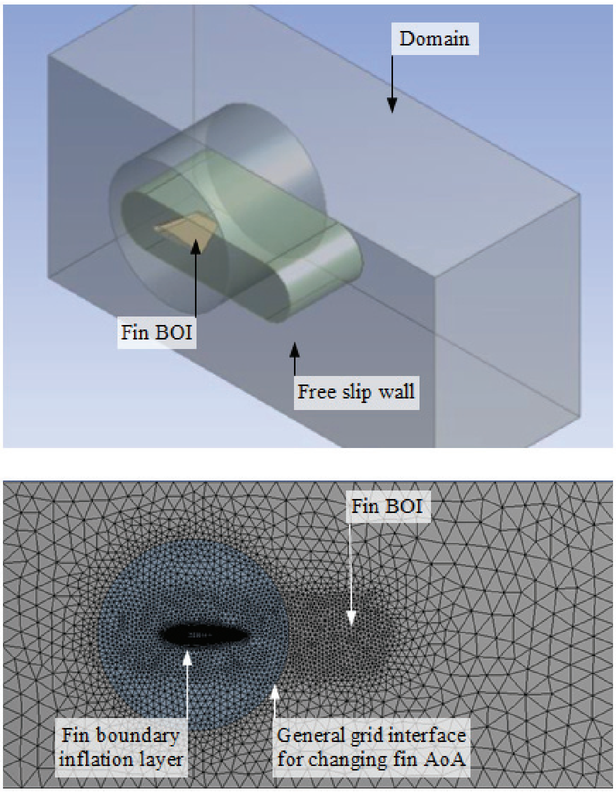


Figure 4: Free stream CFD model and corresponding 0.8×10^6 element mesh.

The "fins on hull" CFD model and mesh is shown in Figure 5. The hull (with fins) was modelled with a free surface having the domain walls set to TT width (3.55m) and the TT water depth (1.5m). The hull was set to the design draft and the fin AoA set to 15°. A BOI with high density mesh was modelled at the fin, hull and the free surface surrounding the hull to capture all pressure and velocity fields around the fin and hull as well as the resulting wave patterns. A high density mesh was also modelled at the air-water interface to capture the free surface and waves generated by the hull. Considering the symmetry of the hull and fins and appropriate validation, the simulations were conducted using symmetry about the hull centreline plane. Based on mesh uncertainty analysis, a 4×10^6 element mesh was used which gave a maximum uncertainty of 5% for hull lift and 2% for hull drag.

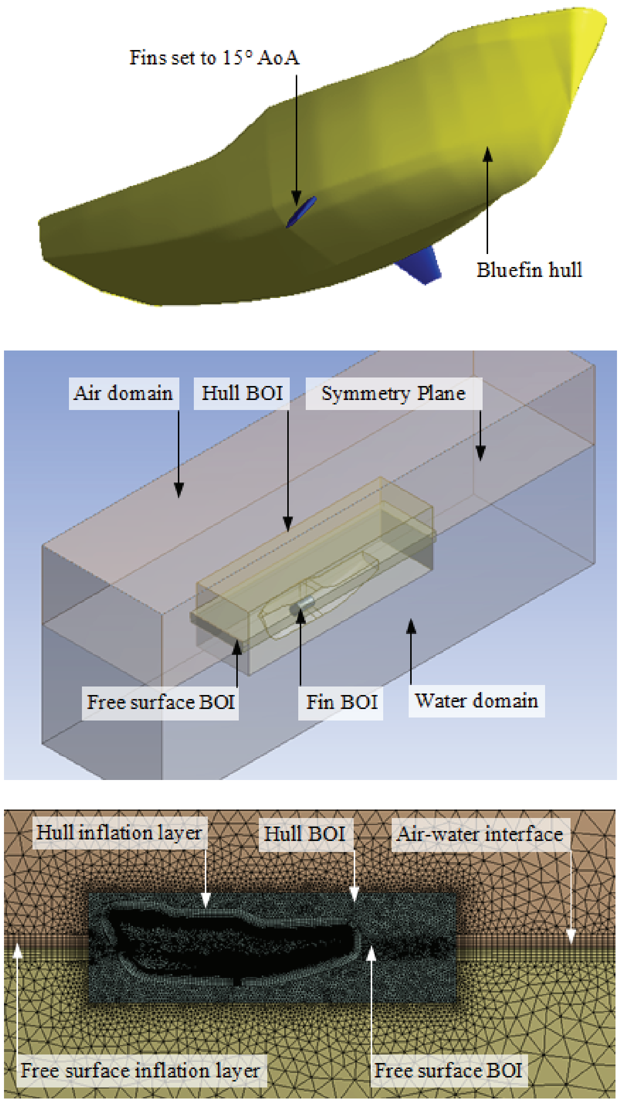


Figure 5: CFD model and corresponding 4×10^6 element mesh used in analysis of bare hull and hull with fins.

5. RESULTS AND DISCUSSIONS

5.1 CFD VALIDATION

The free stream CFD model was validated against the wind tunnel experiment results [4] at Re of 0.931×10^6, as can be seen in Figure 6. The comparison showed a maximum discrepancy of 10% for C_D and 7% for C_L which may be attributed to the boundary layer and turbulence at the gap between the fin root and wall in the wind tunnel experiments. The gap was not modelled in the CFD simulations. Considering the mesh uncertainty presented earlier, the free stream CFD results showed excellent agreement with experimental results.

The CFD predictions of the bare hull simulation were validated against TT experiment results obtained from Condition 2 (shown as Experimental Fluid Dynamics, EFD) and showed reasonable correlation, as can be seen in Figure 7. For direct non-dimensional comparison, the results of both hull lift and hull drag were non-dimensioned as L''_{Hull} and D''_{Hull}, where $L''_{Hull} = L / \rho g \nabla_{Hull}$, $D''_{Hull} = D / \rho g \nabla_{Hull}$ and ∇_{Hull} = hull volumetric displacement (m^3). This comparison also shows a reasonable correlation considering the simplifications made to the CFD model, in particular omission of the skeg and the absence of hull turbulence stimulators both of which were not modelled in CFD.

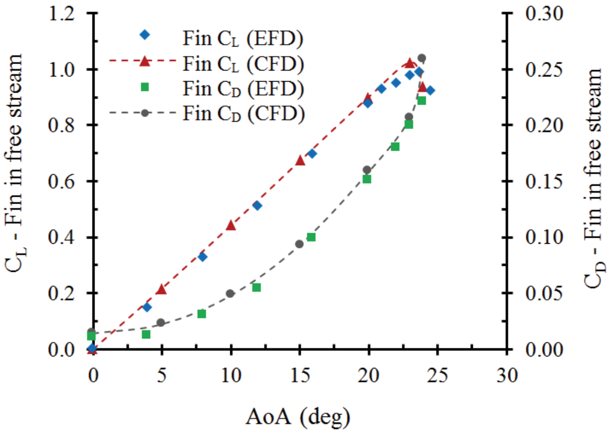


Figure 6: Validation of free stream fin CFD results against wind tunnel experiment results at Re = 0.931×10^6.

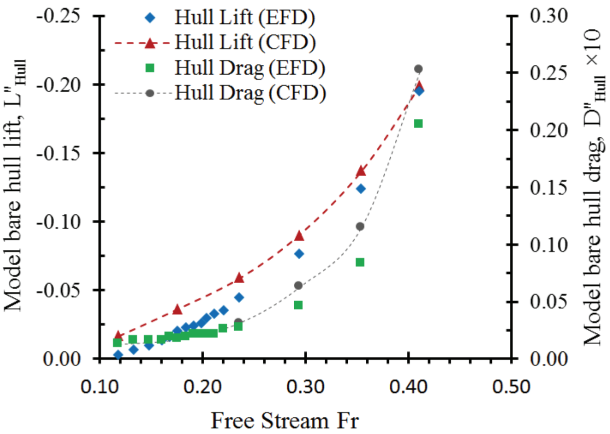


Figure 7: Validation of model scale bare hull CFD results against TT experiment results (Condition 2).

Figure 8 shows the validation for the model fins attached to the model hull. The comparison in Figure 8 showed large discrepancies at low Fr (<0.176) which may be due to the high experimental uncertainty (> ±15%) associated with lower speeds. At Fr > 0.176 however, the CFD and experiment results showed reasonable agreement with a maximum discrepancy of 27% for C_D and 22% for C_L. This discrepancy may also be partially due to the simplifications made to the CFD model. In this case, for the purpose of simplification, the CFD model did not include the gap between the fin root and hull, and hull turbulence stimulators. Additionally, since the experimental values of fin lift and drag are taken as the difference between two separate sets of experiments, the results are prone to higher experimental uncertainties. However, the general trend of CFD results for both bare hull and hull with fins showed reasonable correlation with experiments. Since this case study focuses on relative comparison between different conditions (rather than absolute fin lift and drag), the CFD results were deemed valid.

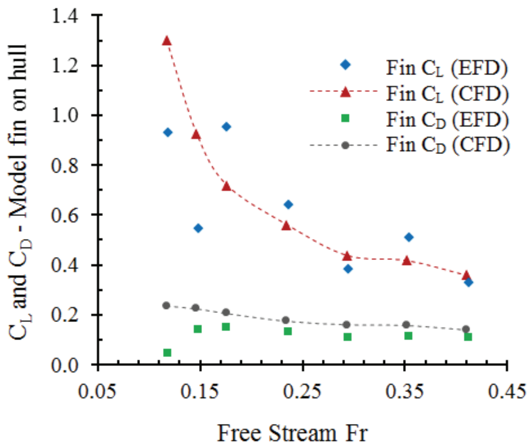


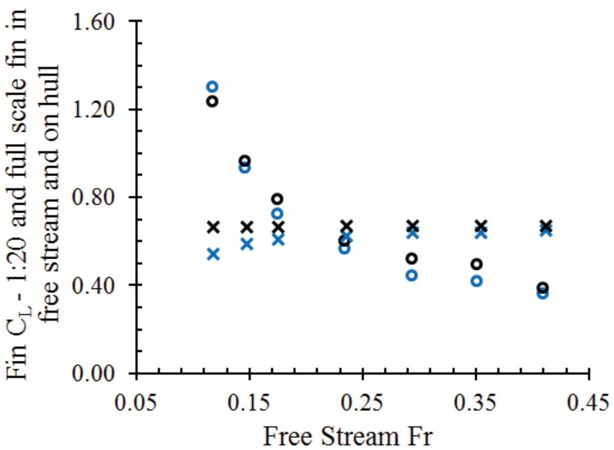
Figure 8: Validation of CFD results for model scale fin on hull against TT experiment results (Condition 3).

5.2 FIN-HULL INTERACTION

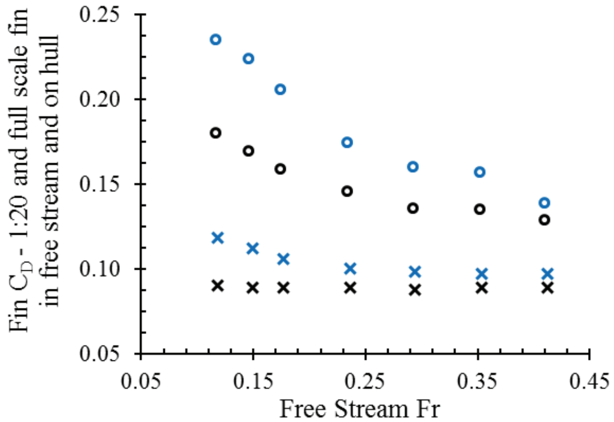
The fin-hull interaction was quantified by analysing CFD results for the performance of the fins (C_L and C_D) when fitted to the hull and comparing it with the free stream fin performance presented earlier. The lift and drag of the fins in TT experiments was determined as the difference between the results obtained for bare hull (Condition 2) and hull with fins (Condition 3) as described earlier. For the CFD analysis, the hull with fins attached was modelled, but the lift and drag was determined as the vertical and horizontal forces acting on the fins only. The fin C_L and C_D was then determined for both CFD and TT model experiments. The C_L and C_D of the 1:20 scale and full scale fin in free stream and when fitted to the hull were plotted against free stream Fr as can be seen in Figure 9a and 9b.

The C_L and C_D of the full scale and 1:20 scale fin on hull were analysed. The difference in model scale and full scale simulations showed discrepancies that generally reduced with Fr. This may be partially due to fin scale effects and correlates with an increase in local fin Re. However, it is important to highlight that in this case, the discrepancy between the free stream results and those for the fins on the hull is also caused by a combined influence of differential pressure and velocity around the hull, fin submergence and variation in hull boundary layer. As discussed earlier, these influencing factors were collectively categorised as fin-hull interaction and the breakdown and individual analysis of these factors was outside the scope of this research.

A more distinct observation from Figure 9 however, was the high C_L and C_D at low Fr and the decrease in both C_L and C_D with increasing Fr when compared to free stream results. In an ideal flow, the C_L and C_D are typically expected to remain constant for changes in velocity, since for a constant fin AoA, both C_L and C_D are a function of the square of inflow velocity (V^2).



(a) Comparison of C_L for 1:20 scale and full scale fin.



(b) Comparison of C_D for 1:20 scale and full scale fin.

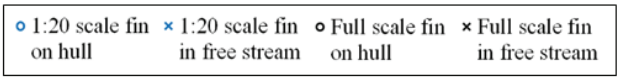
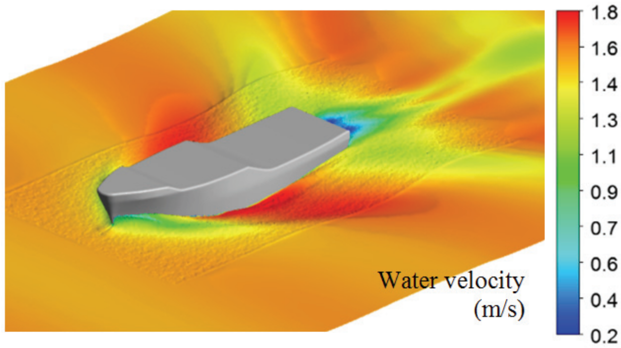
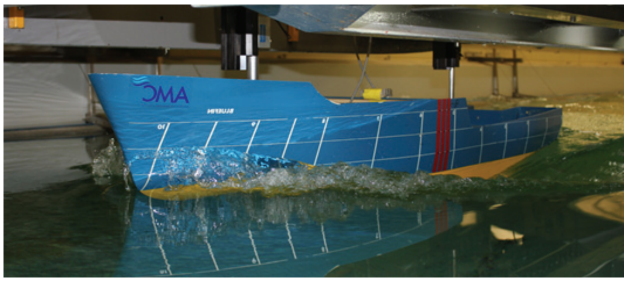


Figure 9: Comparison of C_L and C_D for 1:20 scale and full scale fin, in free stream condition and when fitted to the hull.

This decrease in C_L and C_D for the fin on the hull was further investigated by analysing the flow around the hull and in particular near the fin. As the hull moves through the water, distinct wave patterns (crests and troughs) are formed around the hull [15], known as Kelvin wave patterns and are a function of hull L_{WL} and velocity. Figure 10a shows the CFD prediction of the velocity distribution and wave patterns created around the AMC 96-01 hull which in general form is similar to the actual waves generated by the hull in the TT, as can be seen in Figure 10b. The flow velocity decreases near the bow and stern, thus creating high pressure areas known as the stagnation and pressure recovery regions respectively. There is an increase in flow velocity near midships where the fin is located, as shown in Figure 10a, causing a decrease in pressure in this region and a localised trough.



(a) CFD prediction of wave patterns (at Fr = 0.354).



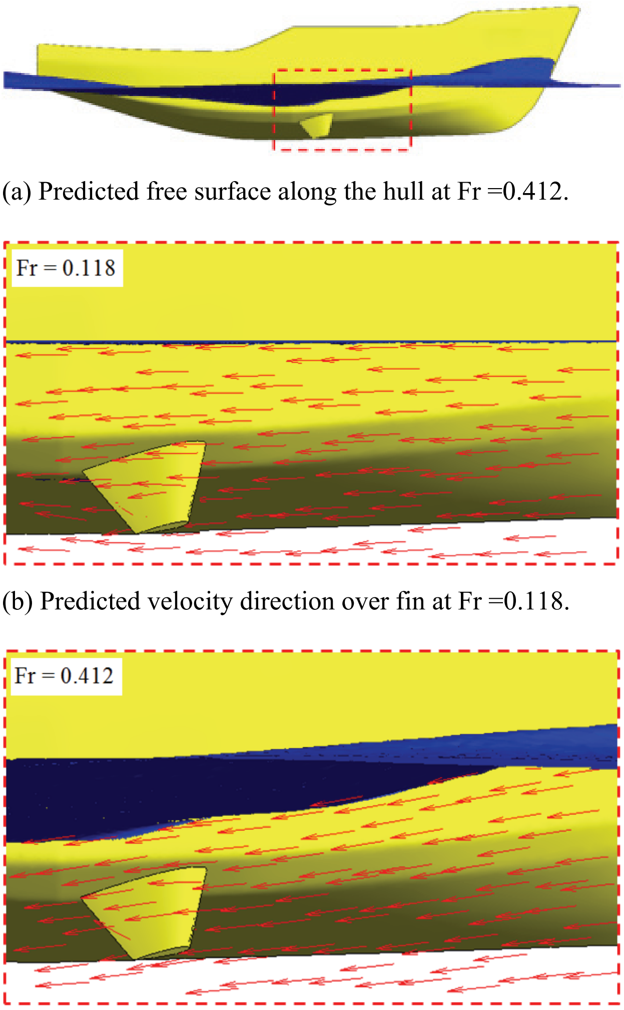
(b) Actual wave patterns in the TT (at Fr = 0.354).

Figure 10: Wave patterns created around the hull in the TT and in CFD simulations at free stream Fr = 0.354.

A profile view of the CFD predicted free surface elevation along the hull at Fr = 0.412 is shown in Figure 11a. Figure 11b and 11c shows normalized velocity vectors plotted on a vertical plane intercepting the fin at mid span. The change in free surface elevation along the length of the hull is associated with a change in the flow direction (from the free stream case) where the fins are located, as shown in Figures 11b and 11c. At low speeds, the wave trough near midships is small (Figure 11b) and has minimal influence on the fin's effective AoA. Hence, this results in a net improvement in fin C_L due to the increased flow velocity over the fin as described earlier. As the speed increases, the wave trough near midships also increases which reduces the effective AoA (Figure 11c) and results in a net decrease in fin C_L and C_D.

Hence, in this case study the variation in flow velocity and flow direction around the hull is seen to influence the fin in two ways when compared to the free stream analysis; 1) increase in lift and drag due to increased velocity in the vicinity of the fin (near midships) as seen in Figure 10a; and 2) decrease in fin lift and drag due to the change in flow direction decreasing the fin effective AoA, as seen in Figures 11b and 11c. The increase in C_L and C_D at low Fr is believed to be a result of the increase in flow velocity over the fin near midships, combined with the effective AoA not being significantly reduced by the change in inflow angle (see Figure 11b). At higher

Fr however, the increase in fin C_L and C_D due to the increase in flow velocity is reversed by a decrease in C_L and C_D due to the change in inflow angle reducing the fin effective AoA (see Figure 11c).



(a) Predicted free surface along the hull at Fr =0.412.

(b) Predicted velocity direction over fin at Fr =0.118.

(c) Predicted velocity direction over fin at Fr =0.412.

Figure 11: CFD prediction illustrating the change in fin effective AoA; showing free surface (blue) and normalized water velocity vectors (red) plotted at a vertical plane intercepting the fin at mid span.

Based on the findings of this case study, it can be seen that the performance of a fin on a hull is dependent upon the hull speed and where it is located within the flow around the hull. The influence of the flow on the fin in this case may not be representative for vessels with larger L_{WL} or situations where the wave pattern generated by the hull is less prominent and has no influence on the stabilisers due to the location or submerged depth of the fins. Also, the influence of the ship motion in a seaway on the flow around the fins should be considered. From a designer's perspective, during preliminary ship design and when determining the longitudinal location and submerged depth of stabilisers, the flow over the hull should be carefully analysed to ensure the flow velocity and flow angle at the fin provide the desired performance to effectively reduce ship motion and minimise fin drag.

Finally, the results presented in Figure 9 were used to quantify fin-hull interaction by determining the percentage difference (or discrepancy) between full scale fin C_L and C_D for the fin in free stream and on the hull. This percentage difference is shown in Figure 12 as a function of free stream Fr.

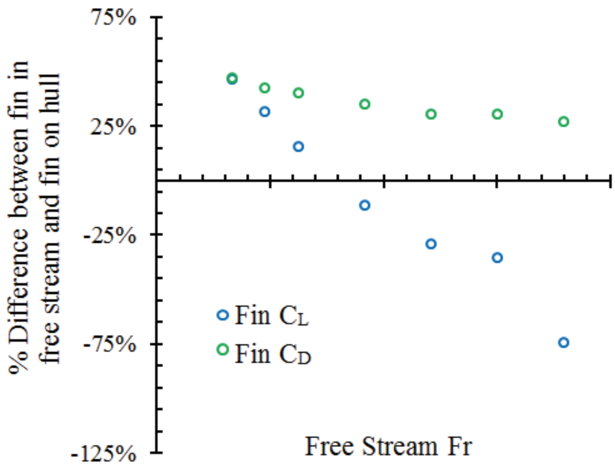


Figure 12. Percentage difference between fin in free stream and when fitted to a hull.

Based on the comparison in Figure 12, it can be seen that using free stream C_L and C_D curves to predict the lift and drag of a fin fitted to the hull can result in considerable discrepancies. In particular for this case study, using free stream data over predicts drag by up to 46% at the lower Fr and under predicts lift by up to 75% at higher Fr. As previously mentioned, this is only valid for the idealized conditions analysed in this case study. In practice, the actual discrepancies will depend on the hull form, fin type, AoA and fin location. That said, this study highlights the possible limitations of using free stream data to predict the lift and drag of fins on a hull. In particular, inaccurate prediction of fin lift and drag could result in a poorly sized stabiliser fin operating at non-optimal AoA. This may lead to insufficient vessel motion reduction, premature stabiliser stall and increase in fin drag which relates to an increase in fuel consumption. Therefore, from an industry perspective, the use of free stream data to design stabilisers may be appropriate as a preliminary estimate, but a more detailed analysis of fin-hull interaction is necessary to ensure accurate stabiliser design.

6. CONCLUSIONS

Understanding and quantifying fin-hull interaction is important in order to analyse accurately the lift and drag of stabiliser fins fitted to a hull. A case study was conducted to study and quantify the fin-hull interaction through a combination of empirical, experimental and numerical methods to investigate the effectiveness of using free stream fin performance data to design a fin stabiliser for a ship hull. In order to isolate the influence of specific variables, fin-hull interaction was quantified by comparing the C_L and C_D values for the full scale fin both in the free stream and when fitted to the hull for a simplified case in calm water. Compared to free stream, the result of the fin on the hull showed high C_L and C_D values at low Fr which decreased at higher Fr. The high C_L and C_D at low Fr were caused by an increase in flow velocity over the fin located near midships. However, at higher speeds, this increase was reversed by a decrease in C_L and C_D due to the decrease in the effective AoA of the fin caused by the change of relative in-flow angle. In this particular case study, using free stream C_L and C_D curves to predict lift and drag of the fins fitted to the hull resulted in an over-prediction of drag by up to 46% and under-prediction of lift by up to 75%. In practice, the actual discrepancies will depend on the hull form, fin type, AoA and fin location. However, this case study highlights the possible limitations of using free stream data to predict the performance of a fin on a hull. From an industry and ship design perspective, these discrepancies may lead to inaccurate prediction of fin lift and drag and hence a poorly sized stabiliser fin operating at non-optimal AoA. This in turn may result in poor motion reduction, premature stabiliser stall, increase in overall vessel drag and an increase in fuel consumption.

7. ACKNOWLEDGEMENTS

We would like to thank the Australian Maritime College for funding the research, T. Lilienthal and L. Honeychurch for assistance in towing tank experiments and Z. Leong for his advice and guidance on CFD.

8. REFERENCES

1. MOLLAND, A.F., and S.R. TURNOCK, Marine rudders and control surfaces - principles, data, design and applications. *Butterworth-Heinemann,* 2007.
2. ABBOTT, I.H., and A.E. VON DOENHOFF, Theory of wing sections - including a summary of air foil data, *Courier Dover Publications*, 1959.
3. SHELDAHL, R.E., and P.C. KLIMAS, Aerodynamic characteristics of seven symmetrical air foil sections through 180-degree angle of attack for use in aerodynamic analysis of vertical axis wind turbines. *Report No. SAND08-2114, Sandia National Labs*, 1981.
4. WHICKER, L., and L. FEHLNER, Free-stream characteristics of a family of low-aspect-ratio, all-movable control surfaces for application to ship design, *Report No. 933, Defense Technical Information Centre*, 1958.
5. GAILLARDE, G, Dynamic Behaviour & Operational Limits of Stabiliser Fins, *IMAM International Maritime Association of the Mediterranean*, 2002.
6. DELLA ROSA, S., M. MACERI, I.M. VIOLA, and S. BARTESAGHI, Design and optimization of a fin stabiliser using CFD codes and optimization algorithm, *16th International Conference of Ship and Shipping Research,* 2009.
7. ITTC 7.5-01-01-01, Recommended Procedures and Guidelines - Model Manufacture - Ship Model. *International Towing Tank Committee*, 2002.
8. MUIR, A., FTV Bluefin - Trim and Stability Handbook, *Alan Muir & Associates*, 2004.
9. MAGGI, A., and G. THOMAS, Full scale ship motions trials on FTV Bluefin, *Report No. IR 95/9. Australian Maritime College, University of Tasmania*, 1995.
10. HIGH SPEED MARINE VEHICLE COMMITTEE, Report of the High Speed Marine Vehicle Committee. *International Towing Tank Conference, Volume 1, p. 306*, 1990
11. LEWANDOWSKI, E.M, The effects of aspect ratio, section shape and Reynolds number on the lift and drag of a series of model control surfaces, *Report No. CG-D-13-94, National Technical Information Service*, 1994.
12. BARLOW, J.B., W.H. RAE, and A. POPE, Low speed wind tunnel testing, *John Wiley & Sons,* 1999.
13. ITTC 7.5-02-02-01. 2002. Recommended Procedures and Guidelines - Testing and Extrapolation Methods - Resistance - Resistance Test, International Towing Tank Committee
14. ITTC 7.5-02-02-02, Recommended Procedures and Guidelines - Testing and Extrapolation Methods - Resistance - Uncertainty Analysis - Example for Resistance Test, *International Towing Tank Committee,* 1999.
15. MOLLAND, A.F., S.R. TURNOCK, and D.A. HUDSON, Ship resistance and propulsion: Practical Estimation of Propulsive Power, *Cambridge University Press*, 2011.

