Carlton, J., Radosavlijevic, D. & Whitworth, S. (2009). Rudder-Propeller-Hull Interaction: The Results of Some Recent Research, In-Service Problems and their Solutions. Paper presented at the First International Symposium on Marine Propulsors smp'09,, June 2009, Trondheim, Norway.

Original citation: Carlton, J., Radosavlijevic, D. & Whitworth, S. (2009). Rudder-Propeller-Hull Interaction: The Results of Some Recent Research, In-Service Problems and their Solutions. Paper presented at the First International Symposium on Marine Propulsors smp'09,, June 2009, Trondheim, Norway.

Permanent City Research Online URL: http://openaccess.city.ac.uk/15022/

Rudder – Propeller – Hull Interaction: The Results of Some Recent Research, In-Service Problems and Their Solutions

John Carlton[1], Dejan Radosavljevic[2], Stewart Whitworth[3]

[1]Global Head of Marine Technology and Investigations, Lloyd's Register
[2]Fluid Dynamics Manager, Lloyd's Register ODS, Technical Investigation & Analysis Consultancy
[3]CFD Specialist, Lloyd's Register ODS, Technical Investigation & Analysis Consultancy

ABSTRACT

The paper describes some recent rudder research that has been carried out by Lloyd's Register that underlines the importance of rudder-propeller-hull interaction in design, both in terms of the flow field properties around the rudder and also the implications for the rudder's contribution to the overall propulsion efficiency. Within this consideration both conventional and variable geometry rudder forms are discussed. Additionally, some recent rudder based problems for a variety of ship types are discussed within the paper together with the lessons learnt.

The paper is based on the results of sea trials, computational fluid dynamics studies and model tests and gives recommendations for the alleviation of such problems in the future.

Keywords

Rudder, Design, Propeller, Drag, Cavitation

1 INTRODUCTION

Lloyd's Register has considerable experience of the rudder-propeller-hull interaction problems and has recently performed a large CFD (Computational Fluid Dynamics) study into the factors that affect the drag experienced by the spade rudders of large container ships [1]. This was prompted by the observation that, under specific circumstances, the rudder can experience a "negative drag"; that is, a thrust which could be useful in reducing the overall drag of the ship.

Apart from the propulsion efficiency aspects of rudder design, cavitation considerations are a concern. High power density propellers need to maintain a balance between operational efficiency and the development radiated hull surface pressures. These also frequently generated strong, cavitating tip and hub vortices which may impinge on the rudders, resulting in paint removal and metal erosion and corrosion. Additionally, the tangential components of the flows shed from the propeller blades have been observed to cause high angles of attack on the rudder with the attendant potential to produce erosive cavitation on the rudder horn and on the blade during normal course-keeping activities of the auto-pilot. Erosive cavitation has also been experienced on the edges, in the horizontal gap between the horn and the blade and behind the horn and pintle bearing housing. Furthermore, the bottom edge of the rudder can also generate erosive cavitation due to abrupt transitions from the rudder leading edge to the base of the rudder, causing edge vortices and bottom sheet cavitation.

2 RUDDER PERFORMANCE RESEARCH

2.1 Objectives

The principal objective of the rudder performance research project was to determine how the drag force on a spade rudder is affected by altering various geometric and flow features. These included varying both the shape of the rudder as a whole and the head-box. Particular attention was paid to cases that exhibit "negative drag" on the rudder as such knowledge.

Additionally, this project aimed to enhance the understanding of flow around the rudder and, in doing so, permit improved rudder designs to be developed.

2.2 Methodology (CFD Model)

The investigation centred on a parent simplified symmetric spade rudder geometry of height 14.4m and was based on a typical design for a large container ship, as shown in **Figure 1**.

Figure 1 – Basic rudder geometry

This rudder geometry, and all subsequent rudder geometries derived from it, are slightly tapered; that is, the cross-sectional area at the top of the rudder is greater than that at the bottom.

Meshes comprising mostly regular, hexahedral cells (of zero skewness) were produced using STAR-CCM+, a commercial CFD code based on the finite volume method

[2]. The mesh incorporated a 10cm thick extrusion layer mesh, consisting of regular prismatic cells orthogonal to the surface of the rudder. The typical size of the final computational mesh was of the order of 1.25 million cells. A horizontal cross-section through a typical mesh is shown in **Figure 2**.

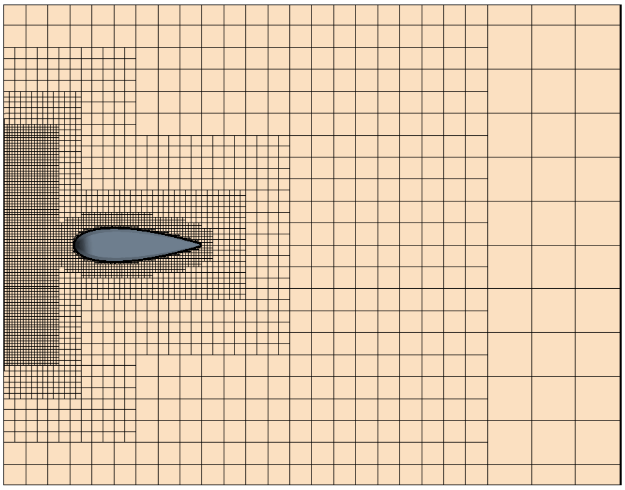


Figure 2 – Slice through a typical computational mesh

Inflow upstream of the rudder was applied either as a uniform axial flow or as idealised wake fields incorporating induced velocities from one of three Wageningen B6.80 propellers. These flows were calculated using PROCAL, a specialised propeller code based on the boundary element method [3]. The characteristics of these propellers are given in **Table 1**.

Table 1 – Propeller inflow characteristics

P/D	0.8	1.0	1.2
Rotation Rate (rpm)	138.5	97.0	75.5
J	0.517	0.738	0.948
K_T	0.195	0.197	0.198
$10K_Q$	0.278	0.346	0.414
Thrust (kN)	6378	3166	1921

A two-equation k-ω SST turbulence model was used to describe the turbulence. The choice of turbulence model was based on a growing body of experience suggesting the k-ω SST model performs better for swirling and separating flows than other eddy-viscosity based two-equation models.

A total of 410 computational analyses were performed as part of this study, with the intention of determining how the drag force on a spade rudder is affected by altering:

- Rudder style
- Rudder shape
- Rudder angle
- Hull angle
- Headbox geometry
- Inflow
- Ship speed

For brevity, only a subset of the results and conclusions are included in the current paper, focusing on those expected to be of greatest interest in rudder design.

2.3 Results and Discussion

Although a wide range of rudder geometries and flow types were investigated, the basic flow features were similar for all cases where an idealised wake field inflow was applied. These are shown in **Figures 3** and **4**. The former shows two high-pressure regions near the leading edge of the rudder at around 70% of the propeller radius, which corresponds to the maximum axial inlet velocity.

The streamlines in **Figure 3** show how the flow past the rudder's leading edge below the centre of the propeller disc passes down the port side of the rudder, while the flow past the rudder's leading edge above the centre of the propeller disc passes down the starboard side. This is due to the tangential velocities generated by the propeller changing the local angle of attack. When compared to the starboard side of the rudder, this leads to higher velocities and lower pressures on the port side of the lower part of the rudder, and lower velocities and higher pressures on the port side of the upper part of the rudder.

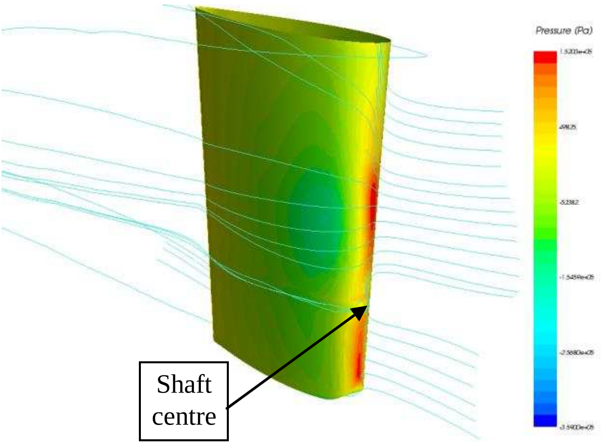


Figure 3 – Pressure contours on rudder with streamlines

Figure 4 shows two stagnation points near the leading edge of the rudder corresponding to the high pressure regions shown in **Figure 3**. A line of flow separation can be seen on the facing (starboard) side of the rudder that begins on the rudder's leading edge at the centre of the propeller disc and moves up the rudder as it goes downstream. A similar line of flow separation occurs on the port side of the rudder, but this one moves down the rudder as it goes downstream.

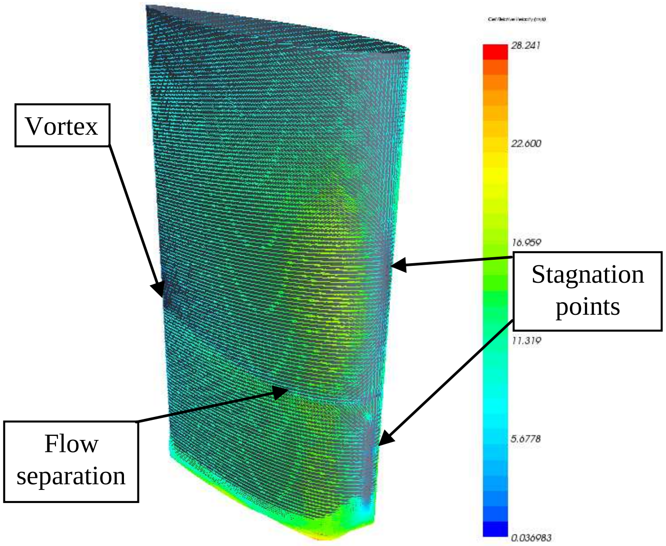


Figure 4 – Velocity vectors around the rudder

Three headbox geometries were examined, denoted here as ‘full’, ‘mid’ and ‘short’, as shown in **Figure 5**.

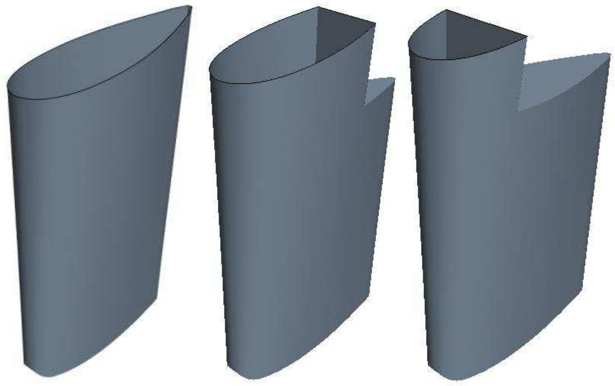


Figure 5 – Rudder with (l-r) full, mid and short headboxes

Given the assumption that the after part of the hull is in contact with the water due to the operating draft and also the influence of the ship’s stern wave, it was observed that although removing part of the headbox reduces the surface area of the rudder, the drag is lowest for those cases with a full headbox. This is because, for those rudders with a mid or short headbox, unstable flow develops in the stagnation region behind the headbox and this increases the drag forces far more than having a smaller rudder surface area reduces them. For example, a typical case setup yielded rudder drag forces of 27kN, 70kN and 246kN for the full, mid and short headboxes, respectively. The snapshot from the animation presented in **Figure 6** shows the potential severity of the unstable flow that can develop behind the short headbox configuration.

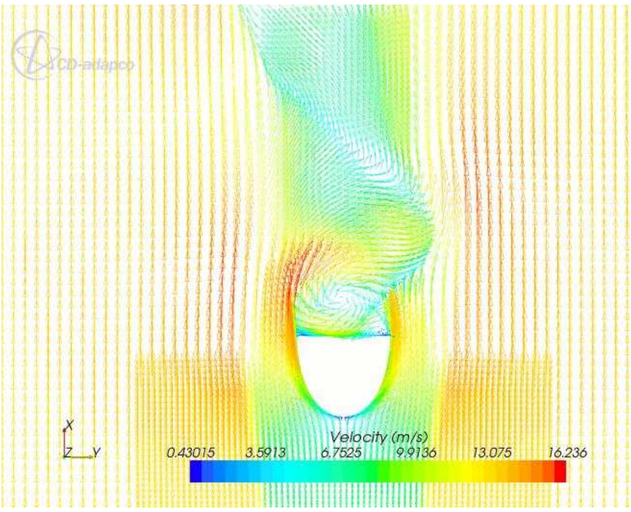


Figure 6 – Velocity vectors showing unstable flow behind short headbox (on conference CD, click picture to animate)

Another factor that can significantly affect overall rudder drag is the angle that the hull above the rudder makes to the horizontal. Increasing this angle leads to reduced rudder drag. This is thought to be due to the variation in pressure on the leading edge of the rudder, which can be seen in **Figure 7**. In **Figure 7**, the angle between the hull and the horizontal decreases from left to right, corresponding to an increase in the maximum pressure on the tip of the rudder. This phenomenon occurs because the rudders in those cases with an angled hull are effectively situated in an expansion, which leads to lower pressures.

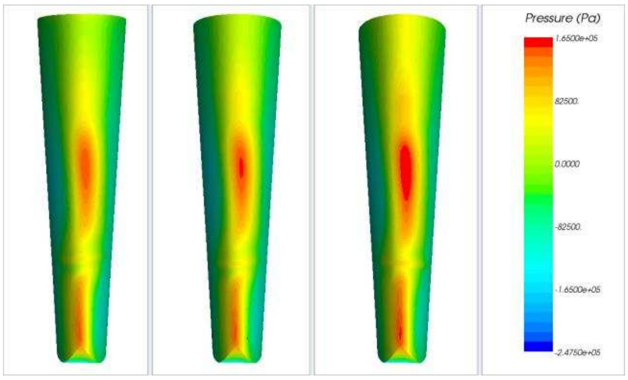


Figure 7 - Typical pressures on leading edge of rudder with hull angle (l-r) 10º, 5º and 0º

When investigating the effect on drag of the shape of the rudder a large number of modifications to the rudder geometry were tested. The starting point for these modifications were the basic rudder; shown in **Figure 1**. Two of the modifications to the basic rudder involved tapering and stretching, as follows:

- Tapering - increasing both upper chord length and upper thickness by a certain factor (1.1, 1.25 or 1.4) while simultaneously decreasing the lower chord length and lower thickness by the same factor.
- Stretching - increasing only the upper chord length by a certain factor (1.1, 1.25 or 1.4) whilst

simultaneously decreasing only the lower chord length by the same factor.

Shape adjustments were performed in this way to attempt to minimise the change in rudder surface area caused by the change in shape. Example cross-sections are shown in **Figure 8**.

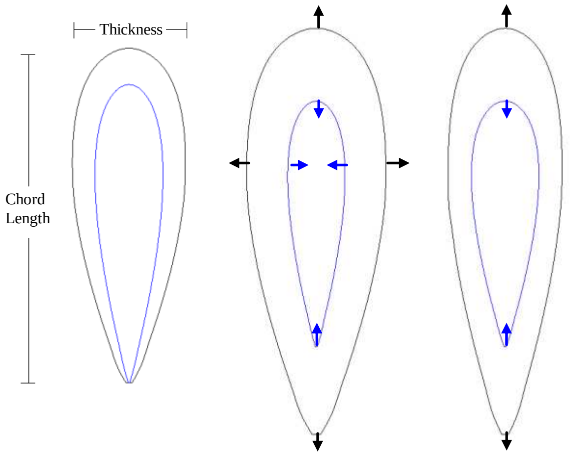


Figure 8 – Upper (black) and lower (blue) cross-sections of (l-r) Basic, Tapered 1.25 and Stretched 1.25 rudders. Arrows indicate the modification compared to the Basic rudder

Increasing the tapering or stretching of the rudder reduces the rudder drag, caused negative drag forces to be observed for the more highly tapered or stretched rudders with certain inflow conditions. Little difference was observed between rudders tapered and stretched by the same factor.

The main reason that tapering and stretching causes a reduction to the rudder drag is that, although the rudder surface area is kept roughly constant, the proportion of the rudder surface that is in the slipstream of the propeller is reduced. This leads to lower pressures both on the rudder's leading edge and on the lower port side of the rudder. Note that, although tapering or stretching the rudder in this way causes a desirable reduction in the drag, it may also reduce the effectiveness of the rudder.

The findings from the tapering and stretching tests were then used to apply two further sets of modifications, the "Longback" and "Thinned" rudders. The latter were created by reducing the thickness of the upper and lower cross sections of the basic rudder shape by 50% and 75%. These two designs were designated the "Thinned 0.5" and "Thinned 0.25" rudders, respectively.

The "Longback" rudders were so called because they were lengthened by stretching the rudder aft of the widest point by a certain factor (1.1, 1.25 or 1.4) whilst leaving the length fore of the widest point unchanged. This operation was performed on both the upper and lower cross sections, both of which also had their thickness reduced by the same factor, as shown in **Figure 9**.

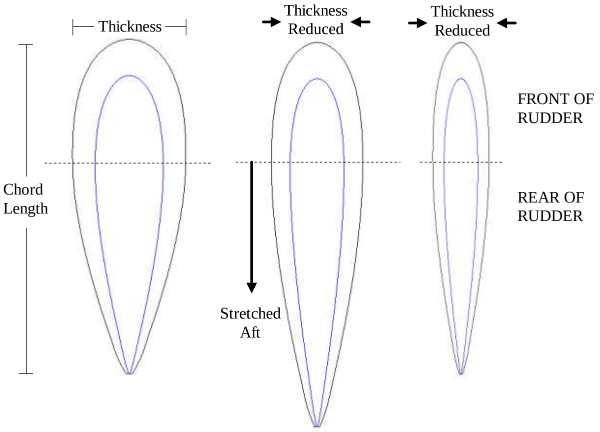


Figure 9 – Upper (black) and lower (blue) cross-sections of (l-r) Basic, Longback 1.25 and Thinned 0.5 rudders

Typical drag results for the various rudder shapes discussed are shown in **Figure 10**.

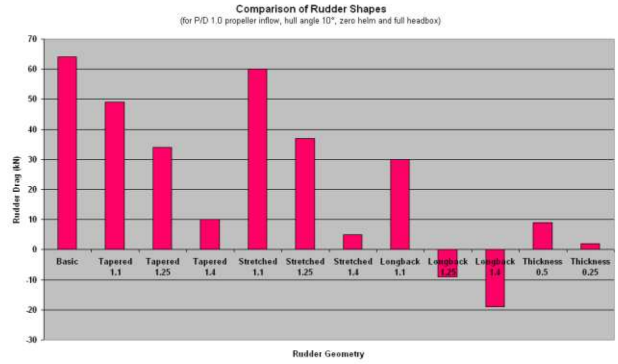


Figure 10 - Typical drags for rudder shapes

The drag-reducing effect of increasing the tapering and stretching of the rudder is clear. However, it is the Longback rudders that experience the lowest drag, in some cases so low as to be negative; that is, the rudder experiences a thrust. One adverse point that arises from this approach to rudder modification is that the increased length of the rudder is likely to lead to much higher drag forces during turning manoeuvres, potentially leading to structural problems. The two "Thinned" rudders were created with this in mind as they have the same length as the basic shape rudder. Although a significant improvement was seen in the drag values compared to the basic rudder, the low thickness of these two rudders could cause structural problems and increased cavitation due to the sharpness of the leading edge.

More detailed analysis of the drag force components reveals that, for a rudder of particular area, changes in the force are almost entirely due to changes to the pressure forces on the rudder. The shear forces remain almost constant, as shown in **Figure 11**. This suggests that rudder surface area is not as significant a consideration when evaluating rudder drag as the shape of the rudder, and that, therefore, rudder drag minimisation need not adversely affect the manoeuvring capability of the rudder.

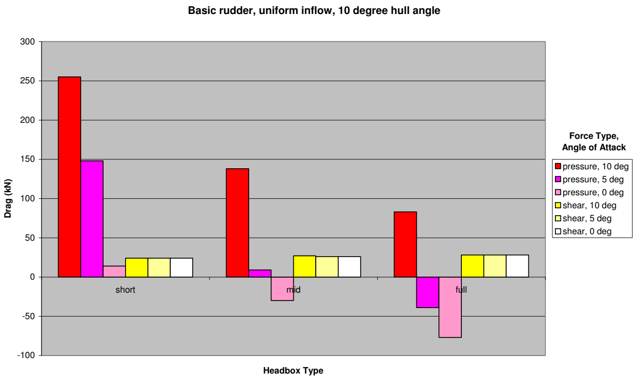


Figure 11 – Pressure and shear components of drag force for the three headbox variants of the basic rudder at various rudder angles

Consideration of the flow field generated by the propeller is crucial as different inflow conditions produce vastly different pressure profiles on the rudder surface and therefore considerable changes in drag. This is illustrated by **Figure 12**, which shows the variation of the drag forces for the "Longback" rudders with inflow type.

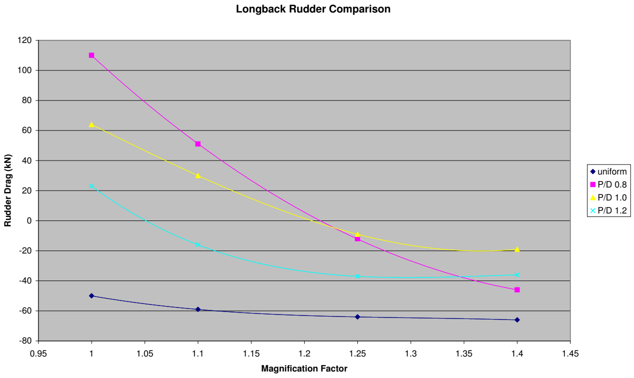


Figure 12 – Variation of "Longback" rudder drag forces with inflow type

All of the rudder design changes that successfully led to reduced, or even negative, drag produce the same effect: an increase in the overall pressure force felt on the rear part of the rudder (i.e. aft of its widest point). This is achieved, variously, by increasing the length of the rear part of the rudder, which leads to a greater rear-facing surface area and, therefore, a greater overall forward component of the pressure force, and by reducing the rudder thickness, which produces smaller low pressure regions on the sides of the rudder that do not extend downstream of the rudder's widest point. Both of these phenomena, which are well demonstrated by **Figure 13**, should be exploited to achieve a rudder that operates with lowest possible drag at zero helm.

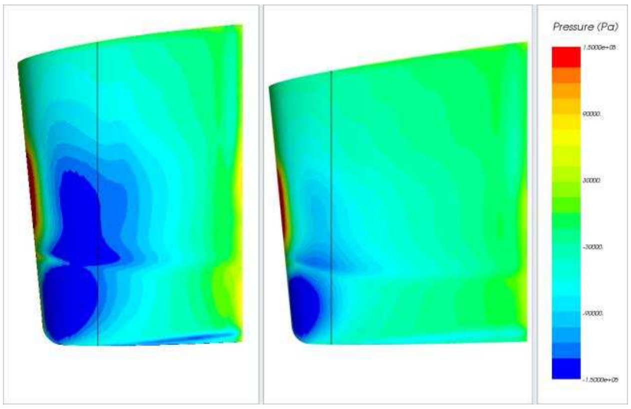


Figure 13 - Pressures on port side of Basic (left) and "Longback 1.4" rudders. The black lines indicate the widest point of the rudder

The reduction in the size of the low pressure region may also reduce the likelihood and extents of cavitation, as shown in **Figure 14**.

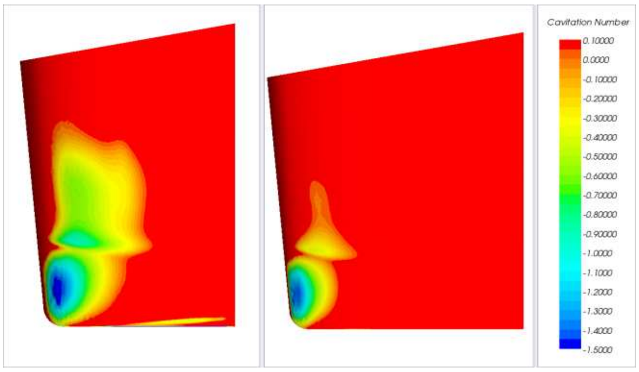


Figure 14 – Cavitation number on port side of Basic (left) and "Longback 1.4" rudders

3 CAVITATION ASPECTS OF DESIGN

Figure 15 shows a general image of a cavitating tip vortex interacting with the rudder, in this case of a small containership, as observed through a boroscope inserted through the hull of the ship.

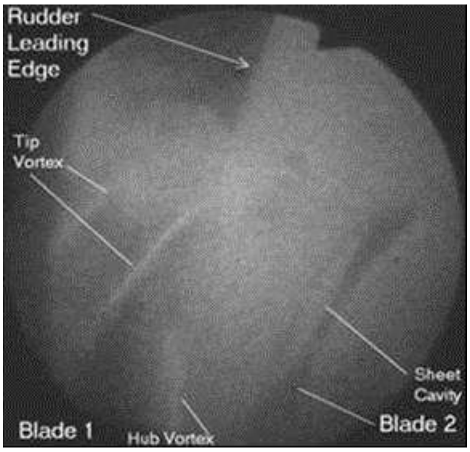


Figure 15 - Image of Cavitation taken through a Boroscope at 200 frames/s.

Erosive cavitation is induced by the way in which the cavitating structures collapse and break up and the energy that is transferred within that process. In the case of propeller-rudder interaction the tip vortex emanating from the propeller blades may pass downstream and then interfere with the rudder in a number of ways. Frequently it is seen that the vortex will rise up the leading edge of the rudder by a certain amount under the interaction of the leading edge and surrounding pressure field. When this happens the vortical structure may be observed to wrap itself around the rudder, or podded propulsor strut, and in some cases form a closed loop vortical structure, **Figure 16**. These types of structure, based on experimental holographic images [4], are thought to comprise systems of micro-bubbles. These bubbles then collapse, probably initiated by a single bubble in the cluster collapsing, in a rapid manner and energy is then transferred to the material surface.

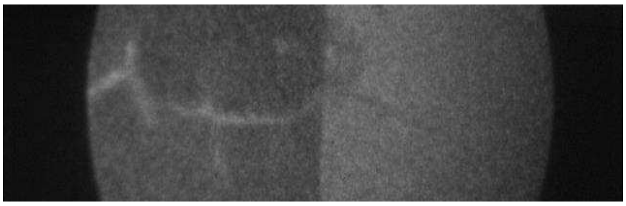
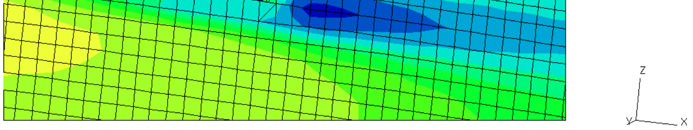


Figure 16. Propeller tip vortex interaction with the leading edge of a podded propulsor as observed through a boroscope.

From Lloyd's Register's model and full scale observations, whenever these discrete ring structures have been observed on either propeller blades or rudders it has always been a precursor to severe erosion being encountered**.** Similar experience has also been reported in hydraulic turbo-machinery practice.

While the presence of cavitation does not necessarily imply erosion, it is true that many rudders fitted to large high powered ships experience erosion. Frequently, attempts have been made to attenuate these erosive effects of cavitation by the application of stainless steel or stellite armour to the rudder and horn: particularly, in the leading edge regions but also on other parts of the rudder. Such attempts, however, have often met with only partial success and have required continuous maintenance during the service life of the ship. There is, nevertheless, some evidence to suggest that more compliant materials may be able to more readily withstand cavitation attack, at least in the more mild cases.

To achieve an acceptable solution for high powered ships a careful design strategy comprising elements of computation and model testing requires implementation. Such a strategy should include the influence that the normal range of auto-pilot rudder angles has on the cavitation dynamics since these angular variations, for the high power density ship designs, will strongly influence the erosion potential of the design. **Figures 17** and **18** seek to demonstrate this aspect in terms of the results of computational fluid dynamics analyses for a rudder pintle area for a rudder angle range of ± 5 degrees.



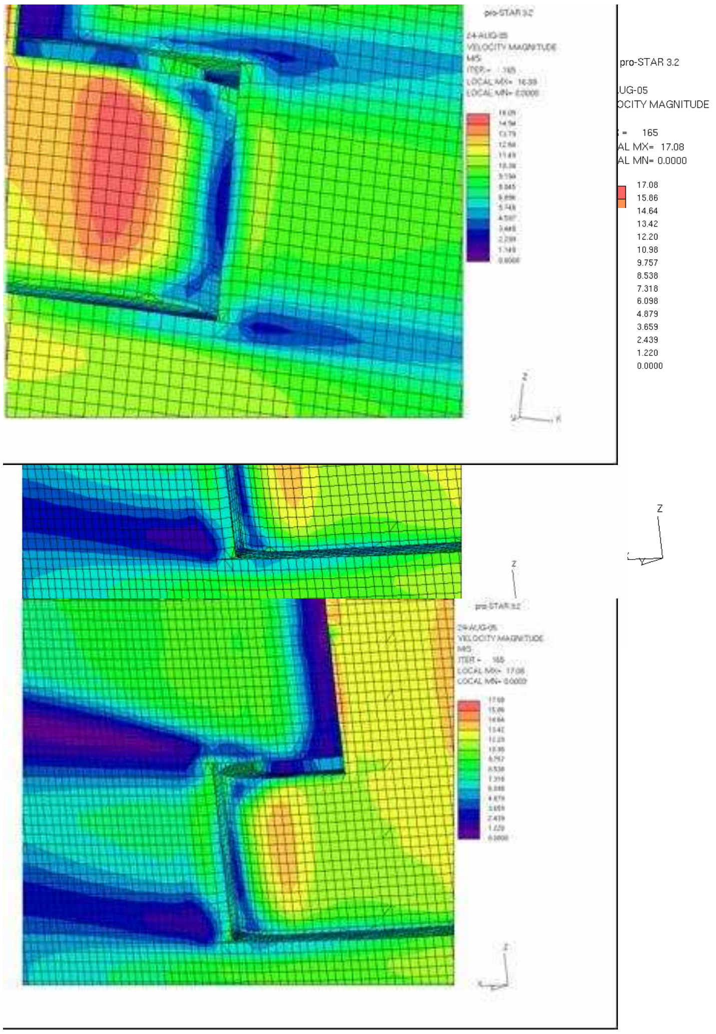


Figure 18. Starboard pintle velocity distribution during a 5 degree turn to port

A number of measures are available for the elimination and partial cure of rudder erosion problems. There is, however, no substitute for undertaking carefully thought out and effective design in the first instance. Indeed, it has been shown [5] that single phase and two phase flow calculations can yield helpful results in this respect at the design stage.

Computational fluid dynamics techniques offer a good potential to reduce the risk of encountering cavitation problems, since the reliability of soft paint techniques when used in cavitation tunnels is not yet as good as similar procedures for propeller blades.

An alternative to the conventional rudder horn-blade configuration is the use of the variable geometry spade rudder concept. This design option allows, in a mean flow sense, for the rotational characteristic of the incident flow from the propeller. Furthermore, for these types of rudders computational fluid dynamic studies have shown good correlation between the predicted actuating torques and bearing bending moments and side forces with the results of model tests.

Recognising that the prediction of erosion at the design stage on rudders is still some way in the future, Lloyd's Register has developed a method for the identification of rudder erosion when the ship is either undertaking sea trials or in service. This method relies on the use of acoustic emission techniques which deploy a number of sensors within the rudder and listen for acoustic signatures from which the location of activity can be determined. When these signatures rise above threshold values, determined by material tests in the laboratory, then erosion can be predicted. Despite the relatively ill-posed nature of the acoustic problem, encouraging results have been obtained as seen by from **Figure 19**. This figure, which is an expansion of the rudder surfaces of a Ro/Ro ship, shows the acoustically predicted extent of the erosion against the actual damage indicated by the sketch lines.

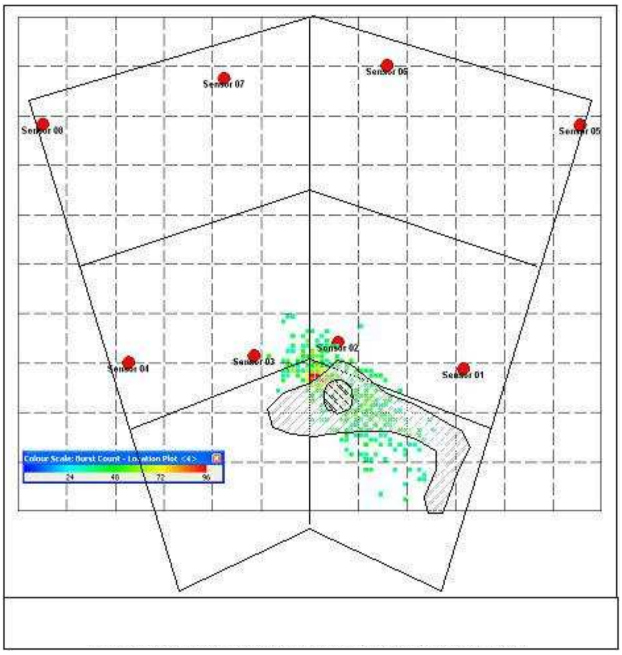


Figure 19. Comparison of an acoustic emission full scale trial prediction of rudder erosion against the actual erosion pattern observed on the ship.

When erosion has been experienced after a ship has entered service there are a number of options available to attenuate the effects of the cavitation.

Stainless Steel Cladding: This technique has been tried as both wide sheets of stainless steel and also as a sequence of adjacent narrow strips of steel. General full scale experience favours the use of narrow strips as the wider strips tend to become detached in service. General experience, however, with this method is mixed and is very dependent upon the severity of the cavitation attack in terms of the energy transferred to the material surface; the quality of welding and the general flow conditions prevailing.

Twisted Rudders; The US Navy developed a design methodology for continuously twisted full-spade rudders and has proven them in service on the Arleigh Burke (DDG51) class of Frigates [6]. The rudder designs were evaluated in the LCC facility and provided a 7 degree increase in cavitation-free envelope at 31 knots. Stepped, twisted rudders have also been introduced to merchant ships and have been found to reduce erosion problems induced by the propeller slipstream and course-keeping operations. These benefits have been seen on both the split blade and continuously curved design types, particularly in the case of container ships and fast Ro/Ro ships.

Scissor Plates: These flat plates are placed in the horizontal gap between the rudder horn and the blade of a semi-balanced rudder. They are particularly effective in controlling the boundary layer within the gap between the blade and the bottom of the pintle housing which can be a source of cavitation development.

Flow Spoilers: Such devices have been advocated for combating erosion on pintle housings and forward facing edges of the rudder blade, immediately behind the rudder horn. There are few reports on their effectiveness and Lloyd's Register's experience with these systems has been inconclusive.

Profiled leading edge transitions between the rudder leading edge and the base of the rudder: Fast vessels should avoid having a 90 degree angle between the rudder leading edge and a flat base plate, since sheet and vortex cavitation have been observed in these regions and have resulted in erosion and corrosion of the base plate within 25% of the rudder chord from the leading edge. Fairings in this region need to be carefully designed and cater for the full range of auto-pilot course keeping angles.

Gaps: These should be as small as practicable between the rudder horn and the moveable blade of semi-balanced rudders. The gap at the base of the horn may be reduced by application of suitable scissor plates. These plates may also be designed to be sacrificial if the erosion is particularly aggressive and may be replaced while the vessel is afloat.

The Annular Gap: This gap, between the aft surface of the horn and the moveable blade, may be reduced in size by fitting vertical strips which block the passage of any flow within this gap. This approach seeks to reduce the cross-flow angle of attack onto the forward facing edges of the rudder blade and has been used to good effect.

4 CONCLUSIONS

The main points emerging from the rudder design research are summarized as follows:

- For all rudder designs tested, drag is minimised when:
    - The headbox is full. Removing parts of the headbox leads to unstable flow that increases drag.

- The angle between the ship hull and the horizontal (water surface) is high, for the cases tested, at least 10°.
- The rudder is at zero helm or within a couple of degrees of it.

- Negative drag is only observed if all of the above conditions are satisfied.
- Rudder drag is predominantly determined by pressure forces. Shear forces are smaller in magnitude and show less variance as conditions change.
- The Tapered and Stretched rudders have reduced drag compared to the Basic rudder because a lower proportion of the rudder is in the slipstream of the propeller.
- Increasing the chord length of the rudder reduces the drag. Increasing its thickness has the opposite effect. Lengthening the rudder's tail (see **Figure 7**) and reducing its thickness is the best technique for reducing drag for all inflow types.
- Consideration of the propeller generated flow field is vital for optimising rudder design with a view to minimising drag.

In the case of cavitation effects, considerable care is needed in attenuating the interaction between the rudder and propeller. This can be approached by:

- In the case of the propeller endeavouring to control the strength of the tip vortex emanating from the propeller.
- Giving careful consideration to the profiling of the leading edge of the rudder, whether this is of a conventional form or one of the twisted leading edge forms now available.
- Using an integrated two phase computational fluid dynamics procedure to assess the cavitation inception on the rudder surfaces. This may involve either a partial hull model used in association with a model test wake field or a complete moddeling of the ship hull. While such studies show areas where cavitation might be anticipated, they do not predict erosion.
- Similarly, cavitation tunnel testing will also show the incidence of cavitation, but are unlikely to predict erosive potential with any degree of reliability.
- The use of acoustic emission methods offer the potential for the early diction of harmful cavitation on rudders at an early stage in the ship's life

.

REFERENCES

1. Whitworth. S. 'Rudder Design Investigation: CFD Assessment' (2007), Lloyd's Register Technical Investigations, MCS/TID/6794.
2. 'STAR-CCM+ 2.08.004 User Guide' (2007), CD-adapco Group.
3. PROCAL V1.0 User Guide (2006), MARIN.
4. Kawanami, Y., Kato, H., Yamaguchi, H., Maeda, M and Nakasumi, S. Inner Structure of Cloud Cavity on a Foil Section. Proc. CAV2001, 2001.
5. Carlton, J.S., Fitzsimmons, P.A., Radosavljevic, D. and Boorsma, A. Factors Influencing Rudder Erosion: Experience of Computational Methods, Model Tests and Full Scale Observations. NAV2006 Conf. Genoa, 2006.
6. Guidelines for the Design of Propellers and Rudders with Respect to Cavitation Erosion. EROCAV Project GRD1-2000-25089.