First International Symposium on Fishing Vessel Energy Efficiency
E-Fishing, Vigo, Spain, May 2010

Flow adapted rudder geometry for energy efficiency improvement on fishing vessels

Alejandro Caldas Collazo, Adrián Sarasquete Fernández

Vicus Desarrollos Tecnológicos, S.L., Vigo, Spain

***Abstract*—One of the most important problems for ship owners of fishing vessels is the fuel consumption. Part of the problem arises from the hull and propeller hydrodynamics. Major modifications of the ship hull for improving its hydrodynamic performance are only a feasible alternative for new ships and the opportunities for improving the performance of old vessels are therefore limited to minor changes. Nevertheless, it is possible to achieve a reduction in fuel consumption through improved hydrodynamics of ship appendages, that applies not only to new designs but also to ships in operation, providing the ship owner with tools for improving the propulsive efficiency of the ship. This article is the continuation of a series of articles about the role of the rudder on the improvement of energy efficiency of fishing vessels. In the previous articles [2 and 7] the studies have been focused exclusively on CFD methodology while in this one towing tank data will be analyzed. The goal of this paper is to demonstrate that it is possible to make reductions on fuel consumption by carrying out slight modifications on the rudder geometry. Designing wake adapted rudders for fishing vessels allows to achieve fuel consumption reductions on service ships with a cheap and simple change (the old rudder must be substituted for the new one) but on the other hand, the rudder needs to be tailor designed for each ship, requiring an extensive hydrodynamic analysis and resulting in somewhat complicated rudder surfaces. In this article, the ongoing studies about energy efficiency in rudders are presented, analyzing two different improved cases. For this investigation, both potential and viscous CFD tools have been used in order to find optimal solutions for the shape of our rudder, using solutions such as the "Costa Bulb", twisted rudders or additional twisted fins.**

Keywords- Rudder, hydrodynamics, energy saving, fishing vessels, CFD.

I. INTRODUCTION

A scenario with oil barrel not cheaper than $ 200 should make us think on the best way to adapt our economy and industries to this situation, that will probably be a reality in a few years.

This scenario has a huge impact on the industry but particularly affects maritime transport and fisheries sectors in which the cost of fuel is the most significant expenditure having a direct impact on the income statement. Every ship owner is nowadays interested on every technology able to reduce the fuel bill, devices providing small improvements in performance and which were despised a few years ago became now very interesting.

Since the construction of the first ships, they have always been equipped with mobile devices for the government, having evolved from primitive paddles arranged on the sides of the ship to the conventional blades attached to a rotating vertical shaft located on the ship's stern. Stern propellers are the most common propulsive solution nowadays; this arrangement creates a flow of water at high speed, which impacts on the rudder surface, improving this way its manoeuvring response.

As the efficiency of a typical propeller can be about 60%, we wonder whether it is possible to increase the propulsive performance of the ship trying to recover some of the energy lost in the propeller so as to improve its interaction with the rudder.

The use of CFD tools together with towing tank testing opens up new possibilities in the study of complex hydrodynamic phenomena occurring in the stern of the ship and in particular between the propeller and rudder. This type of analysis assists the design process of devices like rudders, not only for improved manoeuvring (which is its main role), but also for achieving a maximum energy recovery in the service condition.

II. STATE OF THE ART

As already mentioned, the aim of this publication is to describe and justify the potential of the rudder as an active energy saving device on fishing vessels. First of all we want to classify the ways to carry out an energy saving improving the hydrodynamics of our vessel.

It can be acted in two ways; the first one is reducing energy losses and the second one is recovering this energy losses. For example when we make a good design of the bulbous bow or use some special kind of paint with less friction coefficient, we are making a reduction on energy losses. If the devices as the described in this paper (wake adapted rudders) are used, energy losses from the propulsion system are recovered. This classification is not intended to provide a solid frontier between what reduction and what recovery are but to schematize the sources of energy saving. The propeller is perhaps the biggest source of losses, as consequence, most of the special devices for energy saving are designed thinking on the propeller. There are lots of solutions, each of them providing lots of advantages and disadvantages. A classification of them [3] could be based on the relative position of the device to the propeller:

- The first category corresponded with the one located upstream of the propeller. Systems located here operate in several ways, one being the generation of a rotation in the flow of water from the opposite direction to the turn of the propeller, in order to cancel the rotation of the propeller flow and therefore improving its performance. Other solutions try to improve the axial flow of water reaching certain areas of the propeller, especially the upper region of the disk (such as the Schneekluth flow equalizer duct).
- The second category includes those devices placed in the propeller area. In this area the main focus is the recovery of the rotational energy of the water flow leaving the propeller, like the "Grim Vane Wheel," which consists of a stator located immediately downstream the propeller of the vessel and freely rotating driven by the water flow of the propeller in its outer zone recovering rotational energy in the inner radii.
- This third category contains all those devices located downstream the propeller, mainly appendages normally added to the rudder blade and of course the rudder. Among others we can mention the fins on the Costa bulb type and the "Additional thrust fins" of Ishikawajima Heavy Industries. The rudder is also in this third area, being a major energy recovery device (although its main role is manoeuvring).

Our goal is to increase the recovery energy ratio from the propeller losses since the rudder is located downstream of the propeller. It can be said that three are the main sources of propeller losses; frictional losses, axial losses and rotational losses. Whenever the rudder is placed downstream the propeller, rotational losses are recovered. This is easy to understand taking into account the force diagram exposed in Figure 1 so generally the value of the ruder drag must be minimized or maximized when it points forward. To do this we can act on the rudder in many different ways; one could be modifying the geometry of each horizontal profile of our rudder, adapting it to the velocity field. While other solutions could use devices as Costa bulb type, or employ transversal fins.

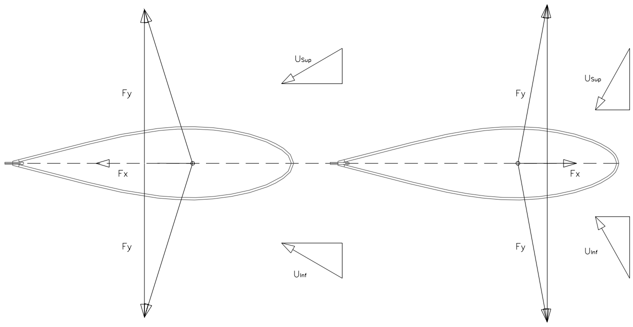


Figure 1. Force distribution on rudder profiles

III. DESIGN METHODOLOGY

The first step on the designing process is the geometry generation of the different rudders. This must be a quick process and of course it must be fully parameterized. As parameters, we decided to choose the main dimensions of the rudder (chord distribution, height, thickness distribution), camber distribution of the different profiles, Costa bulb size and position, size and geometrical distribution of the horizontal fins.

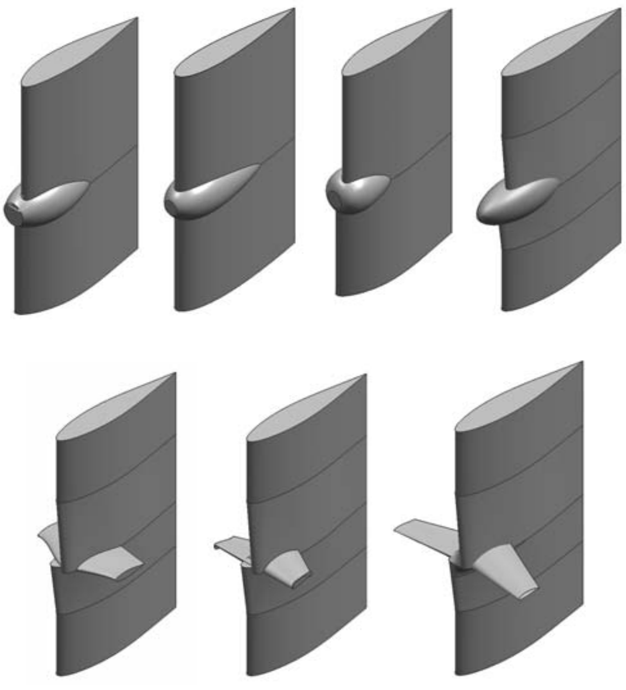


Figure 2. Some evaluated geometries.

A few geometry examples of different rudder configurations are shown in Figure 2.

The generation of the geometries shown in Figure 2 was implemented on the CAD software SIEMENS NX and its generation after the parameterization implementation is immediate.

The mathematical models are defined to be employed in our problem. Our problem is related to a ship moving in calm water, with free surface, a rotating propeller and a rudder behind it. Our goal is to calculate the variation in power for propulsion characteristics with different rudder geometries. We want to carry out this analysis employing CFD, and our starting model are the Navier Stokes equations (N-S) applied to two different phases (air and water). Apart from thinking on the mathematical model, it is necessary to pay attention on the computational effort for the designing process. Solve the complete model described above is very expensive in computational terms so it is not feasible for designing purposes.

The first simplification in the model is to disregard the effects of the free surface; this is not a great error for fishing vessels which normally operate at small Fn. The next step is to break down the whole problem into a series of minor sub problems. The first break is to consider the hull on one hand and the propeller and rudder on the other. There is another break making different calculations for the propeller and the rudder.

The aim of these first simplifications is the definition of the computational domains and the interactions between the sub problems. Talking about the physics, it must be think in another simplifications related to the equations of the mathematical model to be solved. Specifically we must think on how to deal with the diffusive terms in the N-S equations. For example, it is a common practice for the propeller analysis to disregard the viscosity and vorticity, solving potential equations of motion using a Boundary Element Method (BEM). This is not a great error; in fact the integral values (thrust and torque) are in good agreement with experimental results for the propeller. The problem of this approximate model is that the diffusive terms are of great importance for the transport phenomena associated to the fluid flow from the propeller to the rudder and if high accuracy is needed, these effects must be taken into account, not only for the rudder but also for the propeller. Despite of these facts, mathematical models employed for the first designing stage use these simplifications to calculate the flow over the rudder and also a simpler approach by neglecting the diffusive terms.

Besides all this, if the diffusive terms are modeled we must deal with the modelling of turbulence. For this case the choice is to employ Reynolds Averaged Navier Stokes Equations, coupled with a two equation turbulence model (such as k-e or k-w) and with a wall law.

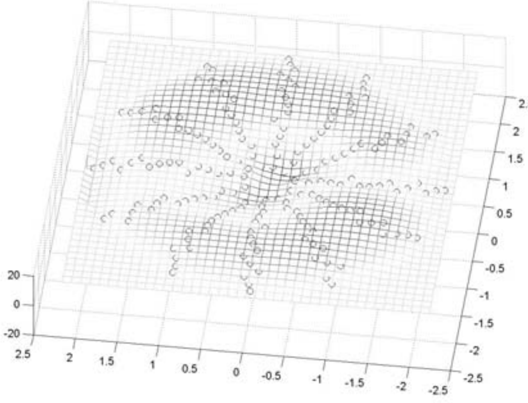
Three different approximated models have been employed:

- RANSE for propeller and rudder w/o free surface.
- Potential flow for propeller and RANSE for rudder.
- Potential flow for propeller and rudder.

Beyond the mathematical model is discussed the numerical methods employed for our calculations. To solve the potential equations, Boundary Element Method was employed. For RANSE equations the used method was a Finite Volume Method.

Below the methodology of the first two mathematical models will be described. In a first stage, a potential based solver for the propeller and a coupled RANSE model for the rudder have been applied. With this model the aim was try to find a near optimal configuration for the twist of the rudder.

As previously stated, the first stage of our calculations was the propeller operating behind the hull. The water velocities were taken mainly from two different sources, a CFD simulation of the hull or from towing tank measurements. The velocities taken from either of these two sources are used as boundary values for our calculations. For a single screw vessel, these velocities will look like in Figure 3. and Figure 4. The calculated velocities and the interpolated ones needed for the RANSE calculation are shown in Figure 3.



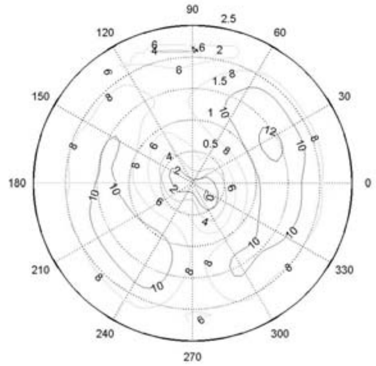


Figure 3. Axial velocity field downstream the propeller

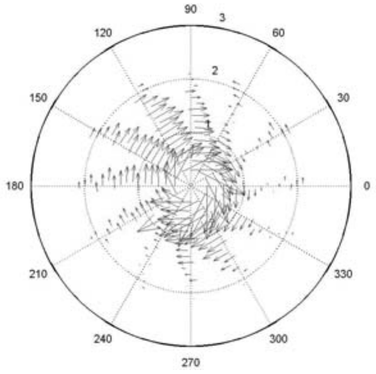


Figure 4. Tangential velocity field downstream the propeller

After the calculations with the propeller we took the velocities downstream the propeller calculated by the panel code as inlet velocities for RANSE calculations. The velocities distribution downstream the propeller and the pressures on the rudder surface are shown in Figure 4 and Figure 5. In these pictures black colour is for higher pressure and white for lower pressure. As can be seen this pressure distribution is motivated by the velocity field and of course it is an asymmetric pressure distribution.

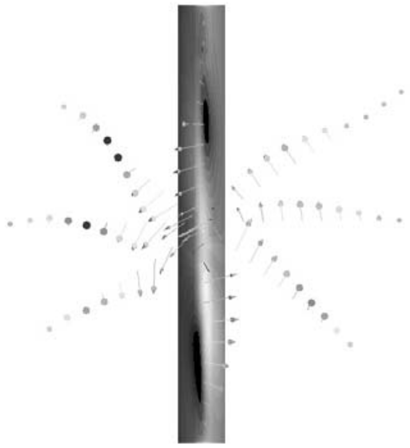


Figure 5. Velocities upstream the rudder and pressure distribution

This is a very complicated flow regime at very high Rn, as have been previously explained and is shown in Figure 6.

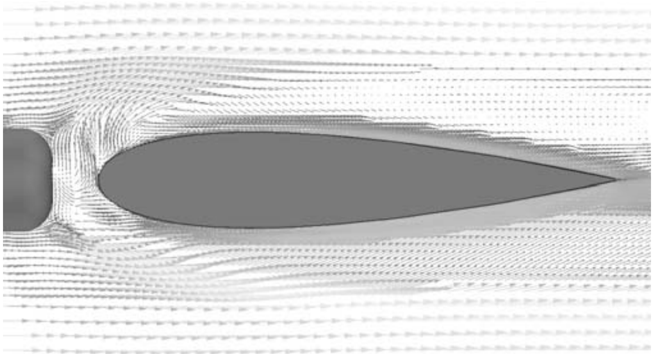


Figure 6. Velocity field around the rudder

In Figures 7 and 8 it can be seen the pressure distributions in two different cases that correspond with the rudders of the second of the towing tank data, calculated with the RANSE propeller-rudder approximation without free surface. This was the method employed at the last design stages.

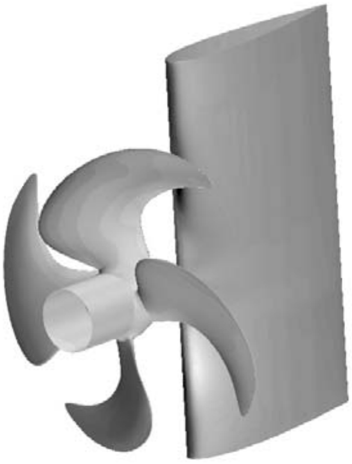


Figure 7. Pressure distribution

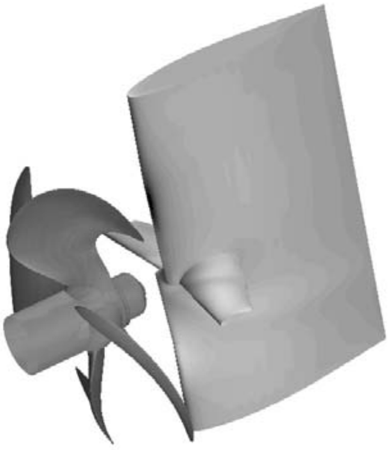


Figure 8. Pressure distribution

There is a fact to be considered when these CFD simulations are carried out. As the simulation is performed with the modified rudder and the energy recovery increases, the rpm of the propeller must decrease to achieve the same ship velocity. The magnitude to be checked is the sum of propeller thrust and rudder longitudinal force. The variation on the propeller thrust at the same rpm between these two cases is very small. Thus the rpm of the propeller must be decreased until reaches the same magnitude. This lead to a situation where the energy recovered by the rudder is less than that calculated for the same rpm, but in any case bigger than the energy recovered by the conventional rudder. There is another fact to take into account, as rpm decrease, of course slightly, J will increase. So the efficiency of the propeller will grow and, at the same time, the suction of the propeller on the hull will be smaller.

Another important aspect to be considered is the scale effect, since two types of simulations were performed:

- Model scale.
- Ship scale.

Model scale simulations were used to validate the mathematical model obtaining good results (spatial, iterative errors). The calculations for full scale showed that there are scale effects not taken into account by the ITTC method for the extrapolation of the model test results. The ITTC method corrects the difference on the Reynolds number between model and ship only by correcting the friction factor. It is a recognized fact that this is not the only effect on the flow, as there are effects as diverse as stern secondary waves, or the well known wake contraction effect when the Rn is increased.

All these effects impinge significantly on the geometry of the stern flow field, were the boundary layer detachments are located. The rudder and the propeller are of course affected by such effects and simulations are the perfect tool to take these disturbances into account. In the simulations it could be seen that the scale effects on the rudder are amplified as Rn increases. This means that the recovery percentage for a well designed rudder is higher for the full scale ship than for the model scale one. For example, for rudders in Figure 7 and Figure 8, the expected improvement employing towing tank data is around 3% for full scale, while employing CFD it can be seen that the improvement is around 5%. This is due to the existence of local Rn higher than the Rn during the towing tank experiments.

IV. TOWING TANK TESTS I

Towing tank testing performed with a 70 m tuna purse seiner with two different rudder geometries is described. The first of the geometries corresponds to a conventional rudder, a NACA 20 profile rudder. The second one is a wake adapted ruder with a Costa bulb designed employing the methodology described before. In Figure 9 the geometries of the conventional rudder and of the twisted rudder with Costa bulb are shown below.

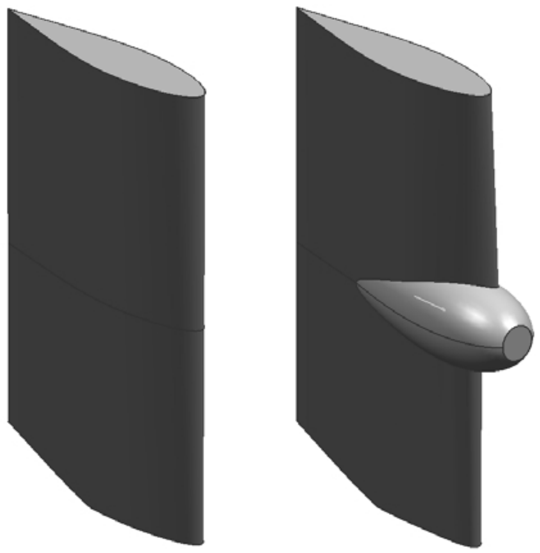


Figure 9. Conventional rudder and twisted rudder with Costa bulb

In the first stage of experiments, rudders were towed at different angles, without propeller or hull, measuring forces in the rudder stock. Positive angles correspond with port side turn and negative angles with starboard turn. The results are represented in Figures 10 and 11. In principle, and viewing these graphs, the behaviour of both rudders are quite similar. The curves are symmetric referred to the angle for the conventional rudder and of course this is not true for the twisted one. For the twisted one at angles below 25º the drag coefficient is lower turning to port side and at angles higher than 35º, drag coefficient is lower when turning starboard side.

Taking a look at the camber distribution at the leading edge, if the rudder is turned to starboard, the profiles are aligned to the velocity field in the lower half of the rudder and misaligned on the upper half. Analysing the camber distribution, it is easy to understand that at low angles the drag is lower when the rudder is turned to starboard.

Generally speaking, the drag is lower (or equal) for the twisted rudder for angles between ±3 to ±25 (of course for 0º is lower for the conventional rudder). The same effect happens for lift coefficient (Cl) up to 15 degrees.

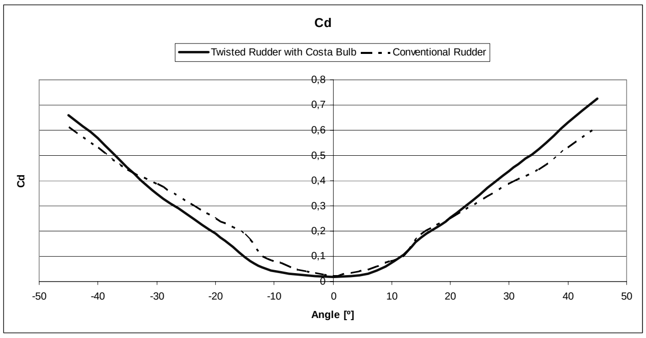


Figure 10. Cd of conventional and twisted Costa bulb rudder alone

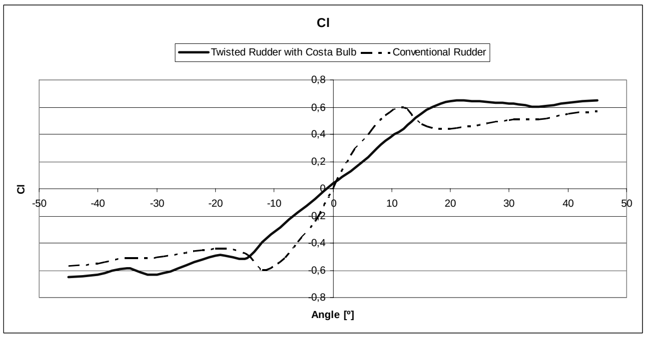


Figure 11. Cl of conventional and twisted Costa bulb rudder alone

In the second stage, trials have been conducted with both rudders behind the hull (without propeller). In Figure 12 and Figure 13, the different values of Cl and Cd measured in the rudder stock are shown. Again the shape of the curves for both rudders is quite similar.

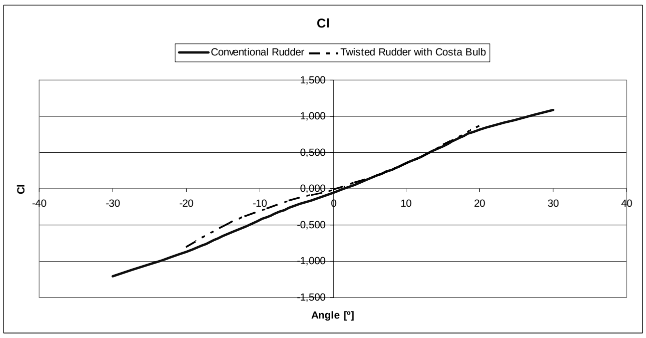


Figure 12. Cl of conventional and twisted Costa bulb rudder on towing condition

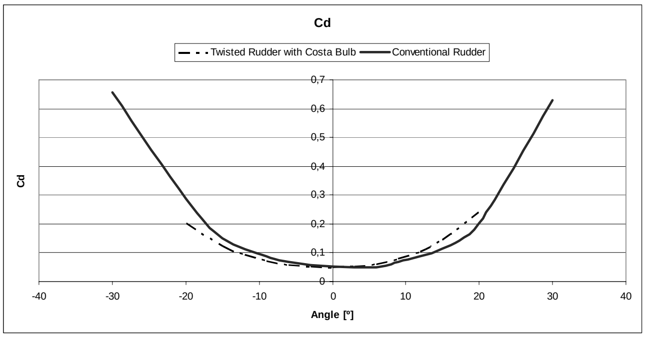


Figure 13. Cd of conventional and twisted Costa bulb rudder on towing condition

In Figure 14 it can be seen the evolution of the towing tank force with both rudders with hull at 0 degrees. For this case the drag force is lower for the twisted rudder as its profiles are better oriented to the flow generated by the hull.

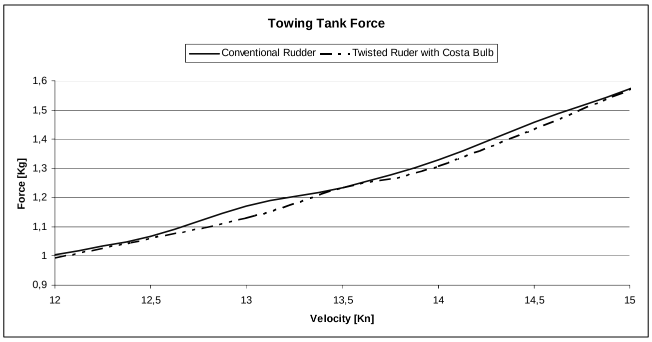


Figure 14. Towing tank force

The next graphs correspond to self-propulsion tests conducted with the two rudders. As it can be seen in Figure 15 there is a reduction in delivered power between 3% and 4%.

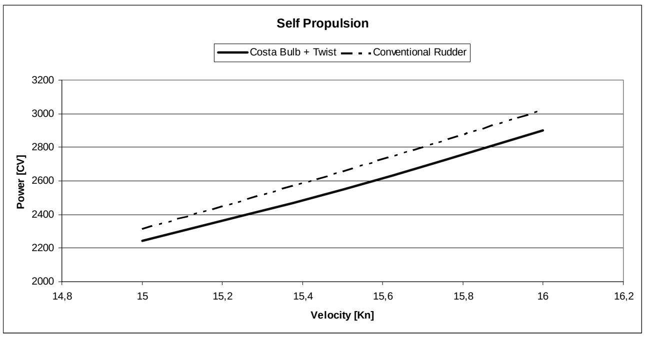


Figure 15. Self propulsion delivered power

Figure 16 and Figure 17 show the values of Cl and Cd of both rudders at different angles.

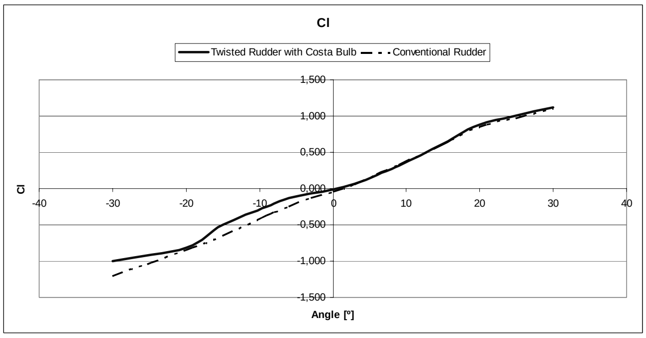


Figure 16. Self propulsion rudderstock Cl measurement

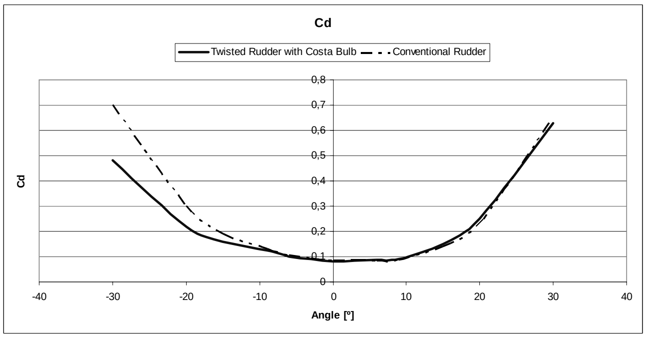


Figure 17. Self propulsion rudderstock Cd measurement

As it was expected, the forces for the twisted rudder are worse turning starboard than for the conventional rudder at port side turning. Even so the difference is not so big.

Figure 18. Twisted rudder designed by VICUSdt fitted on a vessel

V. TOWING TANK TESTS II

Below, the comparison of towing tank data of another tuna purse seiner with two different rudder geometries is shown. The first one is a conventional rudder and the second one is a twisted rudder with two horizontal fins (one on each side).

In Figure 19 it can be seen the towing force of the hull without any rudder. The force is for model scale referred to the Froude scaled ship velocities. The beginning on the main hump is about 18 - 19 kn; and the working velocities are between 15 - 17,5 kn so at first the designed hull forms are whole right.

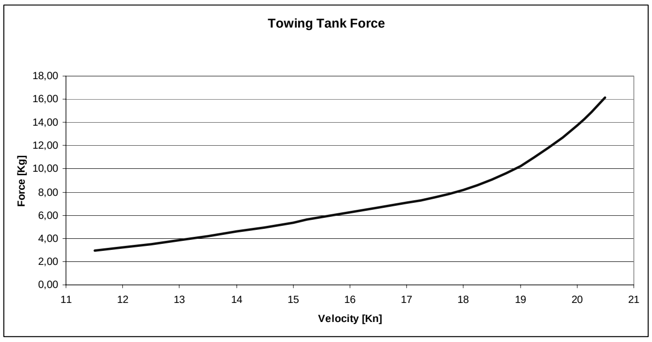


Figure 19. Towing Tank force

After this, the newly designed rudders are fitted, conventional and twisted one, and the towing force is measured for both cases. Figure 20 shows the improvement rate in the towing tank force for these cases, and the hull resistance decreases with the conventional rudder. Here the rudder is also recovering energy from the mass of water that arising from the sides of the hull, by equalizing the flow. There is a worsening in this integral value for the twisted rudder as the orientation of the profiles is quite extreme for this special condition.

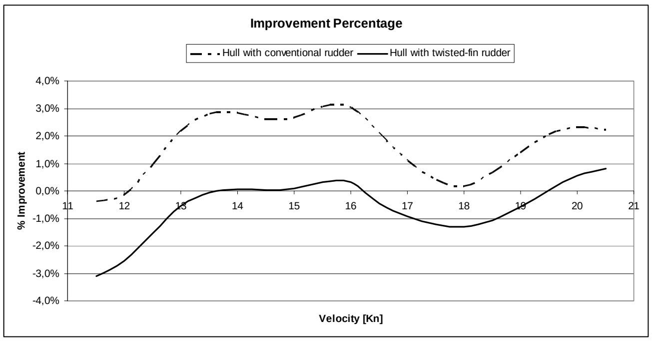


Figure 20. Improvement percentage

There is no problem with this because the twisted rudder was designed for the energy recovery of the rotation losses of the propeller and not to achieve benefits in towing condition.

Figure 21 shows the delivered thrust of the propeller with hull and the conventional rudder. It must be taken into account that for this case too, the rudder is recovering some energy from the propeller.

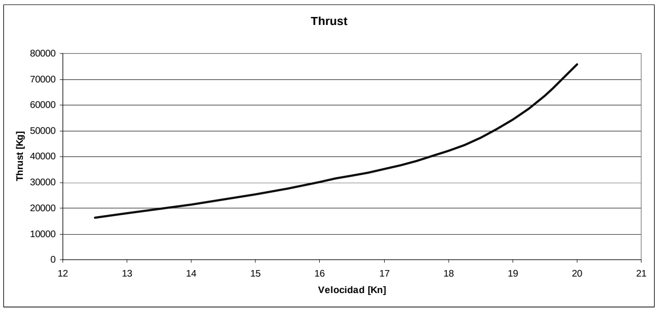


Figure 21. Thrust for conventional rudder case

The goal is to measure the improvement achieved with the twisted rudder so in Figure 22 the improvement percentage of the twisted rudder over the conventional one is showed.

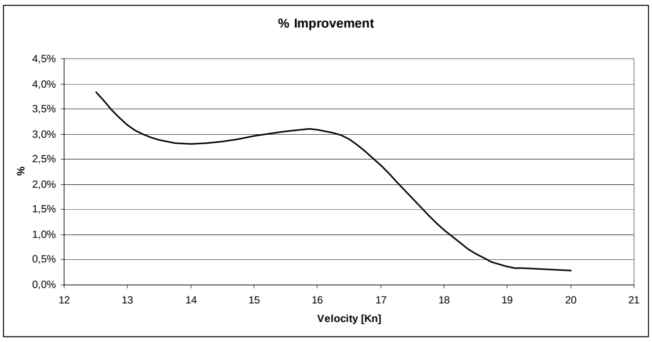


Figure 22. Improvement for twisted rudder

The improvement for a range of velocities from 15 to 17 kn, is between 3% and 2%. Inside this improvement there is a reduction on the drag force of the rudder shown in Figure 23. Transversal force is also lower, Figure 24 and Figure 25, a fact which indicates a better response of our rudder reducing the corrections carried out by the automatic pilot.

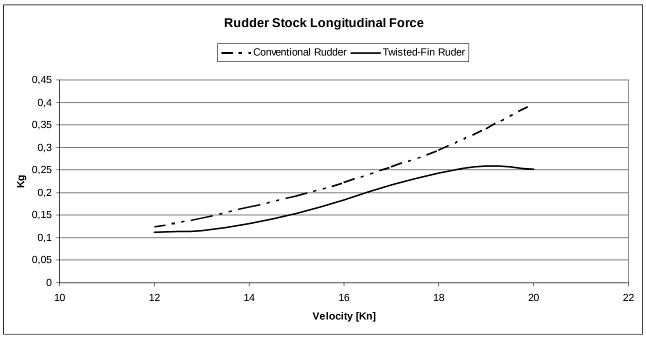


Figure 23. Longitudinal force on the rudder stock

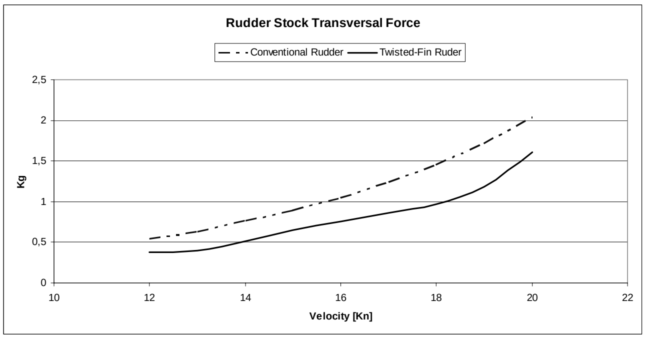


Figure 24. Transversal force on the rudder stock

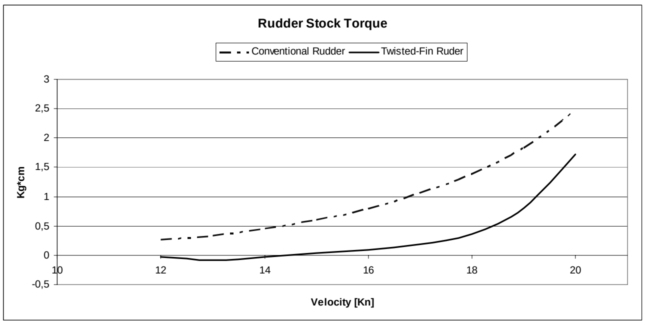


Figure 25. Torque on the rudder stock

Figure 26. Towing tank model photos

As in the previous case the differences between the two rudders (modified and flow adapted) at low speeds are inexistent because of the designing point of the flow adapted rudder and the possibility of energy recovery.

VI. IMPACT ON THE FISHING FLEET

For the impact study on the actual tuna fleet, an estimate of the fuel cost reduction is made using this kind of devices for the second of the analyzed cases, Figure 26.

The ship is a tuna purse seiner with a length between perpendiculars of 74 m, a beam of 14,2 m and a displacement of 4200 t. Typical service conditions for this vessel; assuming an average velocity of 15,5 kn working for 5000 h/year. Besides this, the effect of the generation equipment should be considered because its consume is around 800 kW. Based on the towing tank data for this case, it can be checked that the power required for the ship equipped with the conventional rudder is 2500 kW.

With a fuel price of about 750 $ / t, the annual fuel cost for the ship can be estimated around 1.700.000 $. Considering the efficiency improvement achieved by the twisted rudder, annual fuel savings of 39,000 $ can be expected. The return of investment for the rudder change, based on the figures above, is below two years.

The previous estimation was made taking into account the expected improvement from the traditional extrapolation from the towing tank data. The calculations, as explained before, show that the scale effect depends not only on friction causes but also on the geometry of the flow. The efficiency improvement will be higher than the value resulting if the ITTC extrapolation methods are employed to extrapolate the results to full scale.

Taking our calculations as basis, the annual fuel savings would be bigger, around 60,000$.

VII. CONCLUDING REMARKS.

This study concerns the development of new rudder geometries to achieve better energy efficiency on fishing vessels. A few mathematical models together with the numerical methods used to solve them have been presented with their pros and cons. CFD is presented as a useful tool for the design of these devices as it is possible to make an evaluation of many design alternatives before manufacturing the final design. Towing tank data have been also presented and analyzed for the assessment of the calculations and the applicability of these devices to real ships. This work has been supported in part by the Spanish Ministry of Science and Innovation.

Case 1	
Kt/J^2	*0.25-0.28*
Towing Tank Ratio Improvement for Full scale	*3-4%*
CFD Ratio Improvement for Full scale	*4-5%*
Case 2	
Kt/J^2	*0.21-0.25*
Towing Tank Ratio Improvement for Full scale	*2-3%*
CFD Ratio Improvement for Full scale	*3-4%*

TABLE I. CONCLUDING REMARKS.

REFERENCES

[1] Bertram, V. Practical ship hydrodynamics. Butterwoth-Heinemann. 2002

[2] Caldas, A. and Sarasquete, A. Modification of the Rudder Geometry for Energy Efficiency Improvement on Ships, Numerical Towing Tank Symposium 2009. Cortona

[3] Carlton, J. Marine propellers and Propulsion. 2nd Edition. Butterworth-Heinemann 2007.

[4] Li, Da-Quing. Investigation on Propeller-rudder interaction by numerical methods. PhD thesis. Chalmers University of Technology 2007.

[5] Ferziger, J.H., Perić, M. Computational Methods for fluid dynamics. Springer. 2000

[6] Molland, A. F. and Turnock, S.R. Marine Rudders and Control Surfaces. Butterworth-Heinemann 2007.

[7] Sarasquete, A. and Caldas, A., Investigación sobre la eficiencia energética en timones. Líneas de investigación en Vicus Desarrollos Tecnológicos S.L., 48 Congreso Nacional de Ingeniería Naval 2009.

[8] Söding, H. Limits of potential theory in rudder flow predictions. Twenty-Second Symposium on Naval Hydrodynamics. Washington, D.C. 1998

[9] Stierman E.J. The influence of the rudder on the propulsive performance of ships Part I & II. International Shipbuilding Progress, 36, Nº 407 (1989) pp 303-334 y Nº 408 (1989) pp 405-435

[10] Streckwall H. Rudder Cavitation. Numerical Analysis and Shape Optimization. STG CFD in ship Design. Hamburg. 2007

[11] Tetsuhi Hoshino et Al. Development of high performance stator fin by using advanced panel method. Mitsubishi Heavy Industries Technical Review. Vol 41 Nº6. 2004

[12] Vorhoelter, H., Krueger,S..Optimization of appendages using RANS-CFD Methods, Numerical Towing Tank Symposium 2007.