OPTIMIZATION OF A TRI FOIL STABILIZATION WING FOR AN HSC MULTI HULL USING CFD

Ing. Salvatore Ricca, C.N. Rodriguez S.p.A., Messina,
Ing. Valerio Ruggiero, Researcher, DCIIM, Facoltà di Ingegneria Navale, University of Messina Email: valerio.ruggiero@ingegneria.unime.it
Ing. Filippo Cucinotta, Ph.D. Student, DCIIM, Facoltà di Ingegneria Navale, University of Messina Email: fcucinotta@ingegneria.unime.it
Ing. Andrea Russo, Ph.D. Student, DCIIM, Facoltà di Ingegneria Navale, University of Messina Email: arusso@ingegneria.unime.it

1 INTRODUCTION

The ship of interest is a fast multi hull (obtaining a speed of more than 40 knots) which features a stabilization wing on the fore part between the two hulls.

The wing is composed of two symmetrical profiles connected to the lower part of the cross deck by a vertical support of proper shape and size.

We wanted to investigate the differences in the results between classical theory of the wing sections, used to realize the stabilization wing, and the results achievable with a CFD simulation of the same wing in various conditions.

It is important to optimize not only the profiles of the wings but also the profile of the support for two main reasons. First, it is necessary to reduce the resistance and, secondly, to investigate the absence of cavitation in order to maximize efficiency while guaranteeing the absence of dangerous mechanical stresses on the structure and the materials.

In order to achieve this result, a CFD model of the profile has been constructed and tested in order to check the behavior in various conditions. We also compared the results of the theoretical wing theory with the actual resulting CFD calculations.

After first defining the geometry of the wing, the support, and the flaps, the CFD has been extensively utilized to check the wing performance for a ship without pitch that is moving in a straight route. Desiring more in-depth investigations of the effects of various variables and conditions upon the performance of the wing, the authors performed additional tests involving various scenarios of the typical types of usage for ships. These various tests were important because how the ship is utilized significantly changes the performance of a wing.

Investigative trials have also been carried out concerning ship turning properties. Of particular interest was a scenario where the support of the wings is misaligned with the velocity vector of the flow surrounding the ship. This misalignment results in an incidence angle and thereby generates a “lifting force” on a layer parallel to the surface of the sea. This force can deviate the bow of ship from the intended route.

Additionally, simulations have been performed that investigate the behaviour of the wing concerning pitch movements that can modify the angle of incidence of the fluid on the wing.

The results of this study made it possible to reduce the resistance of the appendage formed by the wings and their support. This reduction of resistance, in turn, led to saving power for propulsion, reducing dangerous vibrations to the structure and saving money in the construction.

2 THE SHIP

In order to better understand the kind of problems studied, it is necessary to give a short description of the ship. The ship is a multi hull ferry, of over 45 mt of length and capable of over 38 knots of speed.

The stabilization system, on the fore end of the hull, is made with a vertical support at midship connected to a horizontal wing. The profile of the wing is symmetrical.

The wing can be oriented, rotating around a vertical axis parallel to the “Y” axis of the ship in order to change the angle of attack. It is also possible to change the position of the flaps on the aft part of the wing in order to have active stabilization of the pitch movements of the ship, using the lift generated.

All the stabilization systems are hydraulically operated.

The goal of our research was to investigate the various ways to achieve optimal performance in a CFD scenario as well as simulating the behaviour of various innovative solutions using a different approach from the traditional way of projecting wing sections from NACA graphic tables.

The wing does not have to generate an important amount of lift as in a hydrofoil, but only enough lift to stabilize the ship; therefore, it is very important to minimize the resistance of the profiles.

(Note: In order to clarify, in this paper we will use the terms Lift and Drag which refer to: Lift: the vertical force generated by the wing, along a vertical axis, Drag: the horizontal force, in the direction of the movement which is a resistance component. This applies when we speak of the wing both with and without the support. When we speak of the only support, the Drag is the force in the direction of the flow according to the theory of wing sections.)
Initially, we decided that in order to obtain a better result we would use a symmetrical profile and then apply CFD to this kind of solution.

We evaluated the flow field around the wing and support in several conditions, basically using as parameters the angles of flaps adopted. We isolated the influence on the whole system of each single element separately: wing, support, and flaps.

Finally, an investigation has been performed to evaluate the possibility of cavitation.

.3 STUDY OF THE VERTICAL SUPPORT

We decided to use CFD to investigate possible innovative solutions for the vertical support. The support can be considered as a vertical wing when the ship is turning or there is a stream or the flow investing the support is such to generate an angle of attack. Under these conditions, the support generates a lifting force that can act to deviate the bow of the ship from her route. We decided to investigate how to minimize this phenomenon, which we referred to as “bow rudder effect” for purposes of clarity.

First, we studied the profile of the support using a 2D approach in order to find the solution with the lowest resistance without “bow rudder effect” while acknowledging the realities of cost effective construction. In order to reduce the undesired lift, we compared the following profiles:

- Full solid support.
- Support with 4, 6, 8 holes
- Support made by 2, 3, 4 profiles one after the other.

The following pictures show some results.

![Fig.1 - Pressure field on the 6 holes profiles with 10° angle of attack.](image1)

![Fig.2 - Flow lines around the 6 holes profile with 10° of angle of attack.](image2)

![Fig.3 - C_D comparison](image3)

![Fig.4 - C_D comparison](image4)

![Fig.5 - C_L comparison](image5)

We decided that the optimum profile was a NACA profile, without holes, because the resistance is still low enough and this solution is easier to realize. In fact, the lift generated by this kind of support (the “bow rudder effect”) begins to be higher than the lift generated by the other profiles (with holes, multiple profiles, etc.) only for...
angles larger than 8°-10°. Furthermore, as we discovered, the lift is reduced by cavitation.

Building on this finding, we decided to test this profile at the variation of the angle of the flow, with a systematic variation in order to have a better understanding of the behavior.

To make the process of simulation easier, we produced a script in Basic to automatically rotate the profile in the 3D-CAD software which input only the minimum angle, the maximum angle and the variation step.

Later, the script generated a mesh for the CFD software according to the different angles of the profile.

At this point, the CFD simulation was started automatically and two loops checked continuously to assure that the calculation was converging. This check verified that the $C_D$ curve was constantly horizontal and, if necessary, changed the values of the $y^+$ parameter.

The final results were imported in Excel for the creation of graphs. The data transfer between Gambit© and Fluent© was done with a “Journal file” while the data transfer between Think Design© and Excel© was done with macros.

A similar proceeding was used to create the CFD simulation of the wing and flaps system.

![Flow chart of the script](image)

**Fig. 6 - Flow chart of the script**

![Volume of calculation, angle of 10°](image)

**Fig. 7 – Volume of calculation, angle of 10°**

![Flow lines around vertical support.](image)

**Fig. 8 – Flow lines around vertical support.**

![Lift and drag, related to a support of length 1 m.](image)

**Fig. 9 – Lift and drag, related to a support of length 1 m.**
.4 STUDY OF THE WING

The wing has a trapezoidal shape, symmetrical with respect to the midship and the wing section is a symmetrical profile type NACA 16.

In order to be able to compare the results obtained by the CFD with the results obtained by the NACA values, we had to test the following cases of simulation:

- Single wing and wing coupled with the support in order to evaluate the effects of the support and the coupling flanges on the flow field.
- Wing of finite length and infinite length, in order to evaluate the edge effects due to end of wing vortexes.
- With and without flaps to evaluate the effects caused by the discontinuity of the lifting surface.

The first of the above simulations was performed comparing the model of only the wing with the model of the wing complete with flaps, coupling flange and support.

It is interesting to note that not only does the support increase the Drag in the “wing only” situation but it also increases the lift.

This increase in the lift occurs for two reasons: 1) the lack of pressure on the upper face of the wing, corresponding to the junction between the support and the wing, and 2) the depression generated in the immediate vicinity which is caused by the local acceleration of the fluid. This result makes a difference between the results obtainable using the values listed on the tables in the NACA profiles (Results taking in account the finite length of the wing but not the presence of the support) and the CFD simulation results. This difference is maximum (100%) for the angle of the wing of 0° and decreases very quickly as the effect of the coupling between the support and wing become negligible compared to the lift generated by the angle of attack. (Note: the angle of the wing is measured from the horizontal)

In the maximum lift situation, with the wing inclined by 5° and the flaps at 10°, the difference is 4%.

Furthermore, we investigated the influence of the edge effects due to the length of the wing.

We created in the model two walls that were parallel to the flow direction, without friction, at each end of the wing. With this solution, the passage of fluid from the higher pressure side to the lower pressure side was blocked and there was no communication between the two sides.

We discovered from the comparison between the two curves that the Drag is basically unchanged in the two situations (the two curves are overlapping each other) but the Lift varies considerably. The effect of the finite length of the wing also cannot be discounted.
Next we created the CFD simulations of the whole wing-flaps-support system, modifying the two degrees of freedom: the angle of the flaps and the angle of the wing.

The range of variation for the wing was $5^\circ$ while the range of variation for the angle of the flaps was $10^\circ$.

The mesh utilized is tetrahedral with about 1.4 millions cells and with an average size of the cells on the wing surface being 8 mm with an increase factor not higher than 20%.

The algorithm for the calculation is a 2 equation k-epsilon model which is suitable for high Reynolds numbers. The laminar sublayer has been computed using a wall function.
Above is the graph of the Drag and Lift at the variation of the Wing angle with the flaps at 0°. It is possible to note that, according to the experimental data, the Lift increases linearly while the Drag has a parabolic growth, as better shown in the following figure, with a different scale factor.

In the same manner, the following graph shows Drag and Lift for a flap angle of 10°.

The following is the graph of the Lift and Drag with a wing angle of 0° and varying the flap angle.

Our final calculations showed that the CFD simulation was producing results that generally matched the theory of the wing sections. However, comparing the CFD results with the results achievable with a classical approach, using the well know experimental parameters, we saw that we have increased the efficiency of the system by adopting a smaller size for the wing which also reduced the resistance while having the same effect on stabilization.
REFERENCES

1. RUGGIERO V., FILARDI V., LAMARI P. (2006), Utilizzo di un codice commerciale di fluidodinamica numerica, per l’analisi degli effetti delle appendici di carena di imbarcazioni da diporto. 2° Convegno SEAMED, 21 luglio 2006, Messina, Italy

2. BAKER Timothy J. (2005), Mesh generation: Art of Science, Progress in aerospace science, Volume 41, Issue 1, January 2005, Pages 29-63


7. RUGGIERO V., FILARDI V., CUCINOTTA F., The influence of the mesh size on the use of a commercial CFD software program for the evaluation of the resistance of the hull of a 42 mt. motoryacht. COMPIT 07, April 2007, Pages 458-466


