CFD Study of an Innovative Catamaran with Asymmetrical Hulls

Antonio Cirello[1], Corrado Damiano[2], Giuseppe Lupo[3], Antonio Mancuso[4], Gabriele Virzì Mariotti[5]

[1]Mi. Me. Srl, Isola delle Femmine, Palermo, Italy; [2]Engineerfreelancer, Palermo, Italy; [3]Ferretti Group, Italy

[4]Dip. Ingegneria Chimica, Gestionale, Informatica, Meccanica, Universita' di Palermo, viale delle Scienze, 90128 Palermo, Italy

[5]Dip. Ingegneria Chimica, Gestionale, Informatica, Meccanica, Universita' di Palermo, viale delle Scienze, 90128 Palermo, Italy

[1]design@doylesails.eu; [2]ing.corradodamiano@libero.it; [3]giuseppe.lupo@ferrettispa.com; [4]antonio.mancuso@unipa.it; [5]gabriele.virzimariotti@unipa.it;

Abstract

The purpose of this paper is the study of the shape of catamaran with asymmetrical hulls by both experimental and numerical viewpoint. The experimental study is based on the towing tank tests on a catamaran hulls shape obtained from previous works, showing that the wave resistance is very low produces a lower energy loss and a greater effectiveness. The hulls shape is optimized by the CFD application by force of commercial software, which shows that the reduction of wasted energy is very sensitive to make some changes; in particular three different width of the catamaran are tested and their performances are compared.

Keywords

CFD; Towing Tank Tests; Catamaran; Motion Resistance; Wave Resistance

Introduction

The rapid development of the high performances ships market, for the commercial and military transport, has motivated in the last decade an increasing interest of designers and ship-owners in the catamarans and in the alternative multi-hulls configurations. They are able to offer a good compromise between demand of speed and seakeeping. In this field, a great attention is devoted to the hulls shape, mainly because the interference between waves hardly affecting the ship resistance (Bertram, 2002; Schneekluthand Bertram, 2002; Larsson and Baba, 1996; White, 1991).

The purpose of this study is the reduction of the energy dissipation due to waves optimizing the section lines of the hull, then the investigation on the possibility of a partial conversion of the wave energy in another suitable form of energy, last but not least, the maximization of the hydrodynamic efficiency of the hull. An innovative form of hulls is proposed and developed for a displacement catamaran, characterized by an "internal" asymmetry of two hulls, given by flat external walls and internal surfaces determining a convergent-divergent canal. The design water line was obtained from previous numerical investigations (Damiano, Lazzara, Mancuso and Virzi' Mariotti, 2003) (Fig. 1). In this case, the energy given to waves does not leave the hull to outside, during the motion of the catamaran, but it is carried in the canal, with an increase of the speed of fluid mass in the first convergent part; and the associated energy is converted into pressure in the divergent part and it is useful for the traction.

Many papers have been published about the resistance of the catamarans with asymmetrical hulls (Bruzzone, Ferrando and Gualeni, 1999; Kaklis and Papanikolau, 1992; Papanikolau, Kaklis and Spanos, 1996; Zaraphontis, Spanos and Papanokolau, 2001); nevertheless, the information on the "internal" asymmetry is really limited. Particularly the purpose of the paper (Bruzzone, Ferrando and Gualeni, 1999) is to investigate the influence of some key parameters on resistance of catamarans, comparing a symmetrical hull with an "external" non-symmetrical hull at the same displacement, in order to gain further insight into a complex problem as the interference between the hulls. In (Kaklis and Papanikolau, 1992) an analytical-numerical method for the calculation of the wave resistance of twin-hull vessels with asymmetrical hulls has been developed and presented by the Authors. An extension of the Michell approach was established in (Papanikolau, Kaklis and Spanos, 1996) for the calculation of the wave resistance of catamarans with asymmetrical hulls. In (Utama, Jamaluddin and Aryawan, 2012) Authors recognize that, while a longitudinally staggered hull design is not immediately practical, their experimental results indicate that as hull separation and stagger are increased, resistance decreases. Paper of (Couser, Molland, Armstrong and Utama, 1997) was directed at developing a fundamental understanding of the resistance components of high-speed transom stern vessel, with particular references to catamarans. Finally in (Zaraphonitis, Spanos and Papanikolau, 2001) the Author showed, in the experimental comparison between "external" and "internal" asymmetry, the greater effectiveness of the configuration, object of the present work, in a certain range of speed, underlining the importance of a deep investigation of this problem.

A process of three-dimensional numerical optimization gave the optimal solution for hull shape, offering an improvement of the previous results (Damiano, Lazzara, Mancuso and Virzi' Mariotti, 2003; Damiano, Stroligo, Virzi' Mariotti and Zotti, 2009). The numerical study of the motion resistance was executed using Fluent CFD code, representing the state of the art among the codes for the modelling and simulation of the fluid flows and the heat transfer. The *Volume-of-Fluid* (VOF) formulation was chosen, since it permitted the numerical modelling of the multiphase systems with two or more fluids exclusive of interpenetrating, founding the tracing of the surfaces of interface between the phases by the method *Interface-capturing*.

Catamaran with Asymmetrical Hulls

A prerogative of convergent-divergent tunnel (Fig. 1) is an acceleration of the water entering into the convergent stroke until the speed $v_c > v_s$; where v_c is the speed of the water inside the tunnel in correspondence of amidships section, and v_s the speed of the ship. So, if a propeller is put inside the tunnel at amidships section, the thrust power, directly proportional to the variation of motion quantity, will be proportional to $v_f - v_i = v_f - v_c$; being v_f the speed astern of the propeller, and v_i the speed of fluid ahead of the propeller.

In the traditional catamarans case, with propeller at the stern of hull, the speed v_i coincides with one v_s of the ship. Therefore, the speed v_f being equal, one has:

$$v_f - v_i = \underbrace{v_f - v_s}_{\substack{\text{traditional}\\ \text{demihulls}}} > \underbrace{v_f - v_c}_{\substack{\text{asymmetric}\\ \text{demihulls}}} \qquad (1)$$

so that the catamaran with asymmetrical hulls requires lower thrust power than the traditional catamaran does. In other words, all the other conditions concerning the advancement being equal, the acceleration of the fluid in the convergent part involves a reduction of motion quantity given to the mass of water by propeller, increasing the catamaran efficiency.

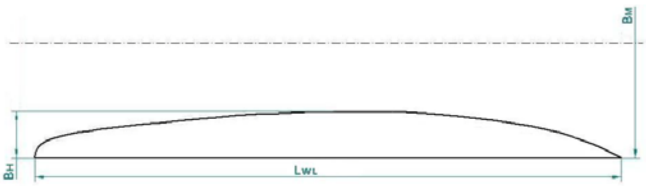

FIG.1 DESIGN WATER LINE

An experimental model was drawn from the following optimized design water line in Fig. 1, with a very simple amidship section characterized by rectangular form (Fig. 2 and 3). The principal geometric parameters of the catamaran are:

length water line	L_{WL} = 1 m;
beam	(table 1);
hull width	B_M = 0,081 m;
draft	T_M = 0,075 m;
displacement	Δ = 9,235 kg;
wetted surface	S = 0,4318 m^2.

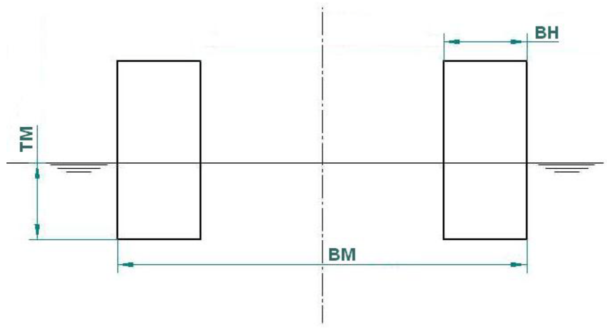

FIG. 2 AMIDSHIPSSECTION [1]

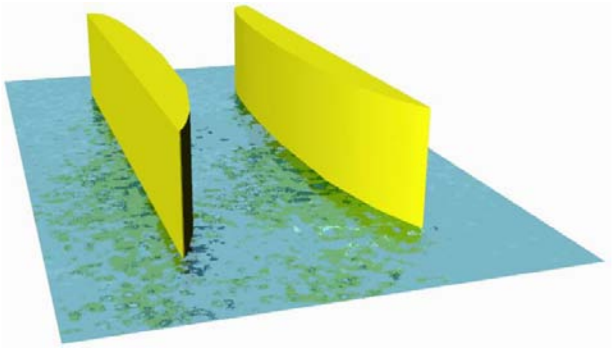

FIG.3 HULLS (RENDERING)

The model was submitted to a series of preliminary towing tank tests (Fig. 4) with the purpose of analyzing the wave patterns produced by models having different values of the ratio between beam B_M and length water line L_{WL} (Fig. 1), which produces useful evaluations for the shape optimization in the subsequent three-dimensional numerical simulations. The examined geometric configurations are reassumed in Table 1.

TABLE 1

Case	B_M/L_{WL}
A	0,340
B	0,400
C	0,440

FIG.4 TOWING TANK TEST

1) *Experimental results*

The ITTC standard procedure for resistance tests consists of the towing of a scale-model at constant speed determining the total resistance R_T. Fig. 6 shows the total resistance curves, versus Froude number, in the cases selected as base for the subsequent numerical simulations.

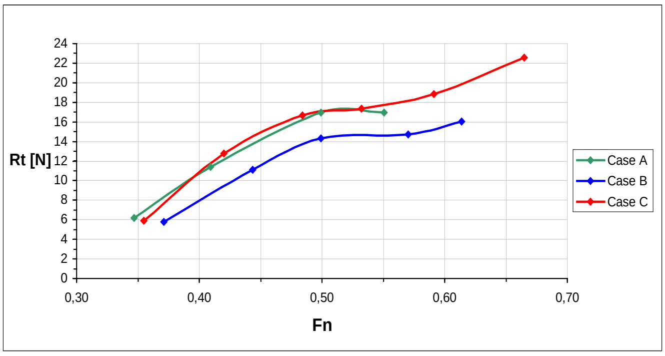

FIG. 5 TOTAL RESISTANCE CURVES (EXPERIMENTAL TESTS)

Froude number, as everybody knows, is defined by:

$$F_n = v_m / \sqrt{gL_m} \tag{2}$$

where v_m is the model speed and L_m the water line length of model. The following relationship permits the drag coefficient calculation:

$$C_D = \frac{R_T}{\frac{1}{2}\rho_m \cdot v_m^2 \cdot S_m} \tag{3}$$

where S_m is the wetted surface. Fig. 6 compares the curves of drag coefficient for the three investigated cases. It is important to observe that, for Froude numbers between 0.45 and 0.50, the curves exhibit the maximum that behaves a progressive reduction of the drag coefficient for greater Froude numbers than 0.50.

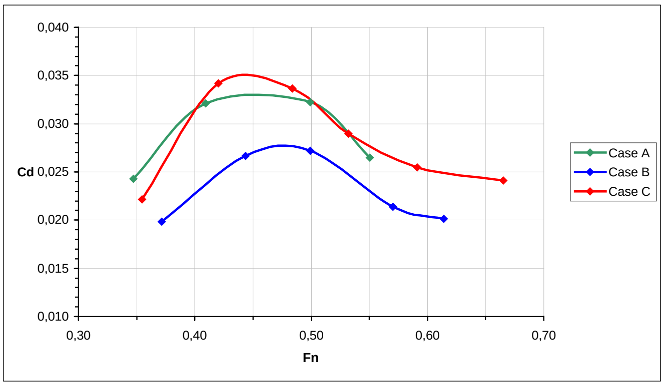

FIG. 6 DRAG COEFFICIENT (EXPERIMENTAL TESTS)

The experimental tests have been done considering a speed range between 0.3 and 0.7 F_n, in order to concentrate the attention on maximum values of drag coefficient. Unfortunately, the limited length of the tank did not permit to investigate on greater speeds than 0.7 F_n.

Numerical Simulations

The preliminary numerical investigations concerned the study on total resistance of a numerical model that reproduces the geometry of the catamaran tested in tank, with the only exception of a light fillet on the inferior edge of the internal surface, as shown in Fig. 1, in order to simplify the blocking strategy of meshing process maintaining the grid to an acceptable quality level. The two surfaces, internal and external, constituting the catamaran's hull have been modelled to the CAD like bi-cubic surfaces by matrixes of 10x6 control points (10 control points for waterlines and 6 for section lines). These surfaces have been saved separately in neutral IGES format.

TABLE 2 DATA OF THE MODEL

Data	Num. mod.	Exp. Mod.
Water line lenght	1 m	1 m
Beam (case B)	0,400 m	0,400 m
Demihullwidht	0,081 m	0,081 m
Draft (T)	0,087 m	0,075 m
Hull underwater volume	0,00923 m^3	0,00927 m^3
Wetted area	0,4131 m^2	0,4318 m^2

Table 2 reports the principal geometric parameters of the preliminary model, compared with the experimental ones. In this table, the different value of draft *T*, between numerical and experimental model, is due to the demand of recovering part of hull underwater volume lost because of light fillet shown in Fig. 7, in order to guarantee a comparison, between numerical and experimental data, at the same displacement.

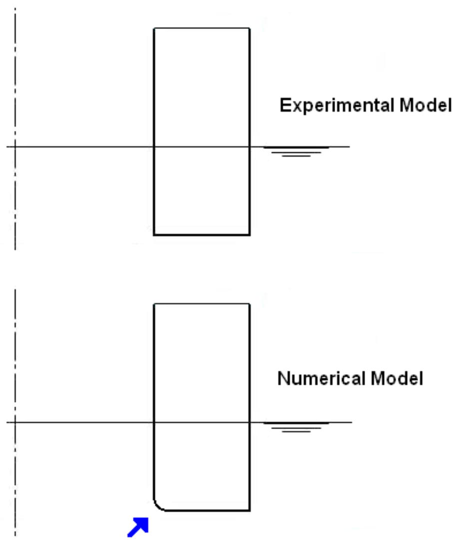

FIG. 7 HULL AMIDSHIPS SECTION

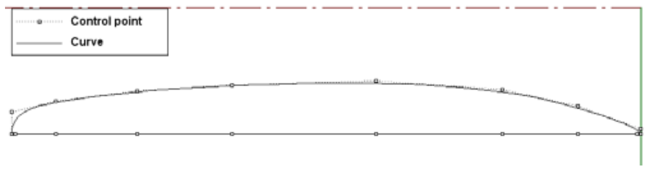

FIG. 8 CONTROL POINTS OF DESIGN WATERLINE

An unstructured hexahedral grid (Fig. 9, 10) has been produced for the numerical investigations. The meshing of physical system is based on the concept of block topology. The definitive version of mesh represents the result of a long refinement process of the blocking strategy with purpose to guarantee a suitable grading of the grid to the fluid-dynamic problem resolution, and to take correctly in account the Prandtl limit layer on the bottom surface of catamaran.

The investigations on the total resistance to motion are executed considering the effects of sea making on free surface and, therefore, the presence of both water and air. Among the problems concerning flows around a hull, the more suitable multiphase model is the *Volume of Fluid* model (VOF). The more suitable turbulence model for towing tank simulations is the *K* - standard; it is based on the Wilcox's theory (Wilkox, 1998) that incorporates the corrections concerning the effects of the lower Reynolds numbers, of compressibility and of cut flows propagation. In Table 3 the principal data of control volume are listed.

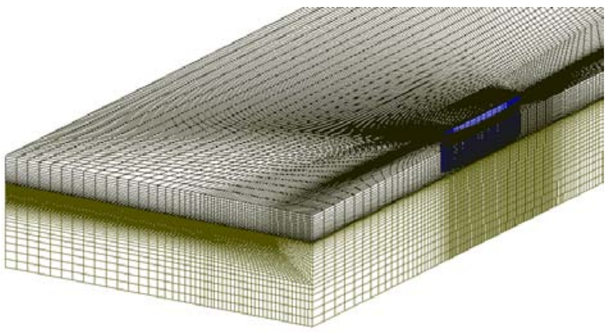

FIG. 9 HEXAHEDRAL GRID FOR THE NUMERICAL INVESTIGATIONS.

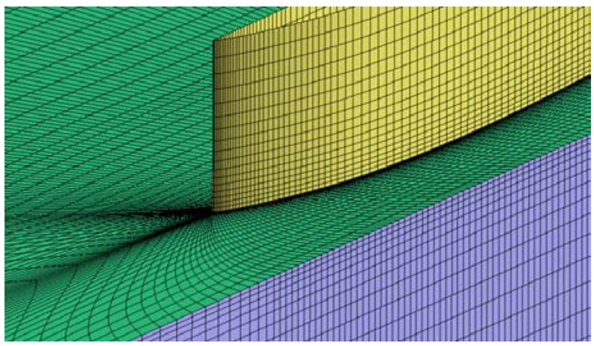

FIG. 10 PARTICULAR OF HEXAHEDRAL GRID

2) *Results of the Preliminary Numerical Simulations*

First series of simulations concerns a range of speed varying between 0.3 and 0.7 F_n; which allows to obtain drag coefficient curves of preliminary numerical catamaran in configuration B with the purpose to show a qualitative comparison between the best experimental results and the numerical ones. Fig. 11 reassumes the results of preliminary numerical investigations.

TABLE 3 DATA OF THE CONTROL VOLUME

Numerical Model	
Water line length L_{WL}	1 m
Total lenght	7 L_{WL}
Lenght ahead of the hull	2 L_{WL}
Length astern of the hull	4 L_{WL}
Width	2,5 L_{WL}
Height	1 L_{WL}
Nodes for element	8
Elements	700304
Nodes	727860
Surface elements *(quads)*	54604
Turbolence model	k-ω standard
Turbolence specification method	Intensity and viscosity ratio
Turbolence intensity	2%
Turbolence viscosity ratio	10
Turbolence kinetic energy	0,000294 m^2/s^2
Specific dissipation rate	2,012686 s^{-1}
Pressure velocity coupling	*SIMPLEX*

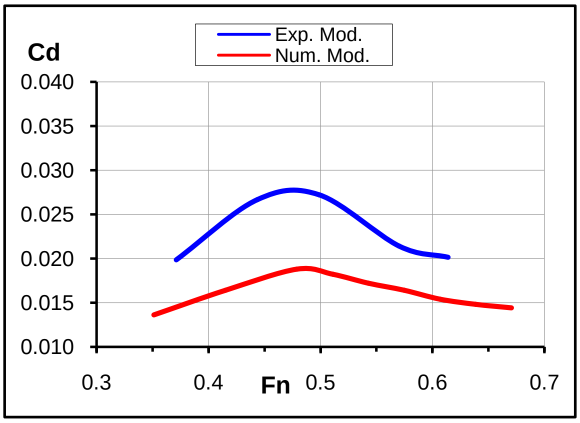

FIG. 11 DRAG COEFFICIENT (NUMERICAL AND EXPERIMENTAL DATA, B=0.400 M)

Both numerical and experimental results give the maximum value of drag in the same interval of Froude number between 0.47 and 0.52, according to (Tuck, Scullen and Lazauskas, 2002) and the Michell's theory (Michell, 1898), subsequently verified by Chapman (Chapman, 1972). The drag coefficient curve provided by CFD solver reproduces, in an acceptable way, the qualitative course of experimental data, although there are non-negligible differences among numerical and experimental values. Such differences are due, first of all, to the limited length of the experimental model because of the little extension of the towing tank; consequently the similitude errors are greater, as in numerical investigations it is not possible to influence a correct simulation upon the turbulence effects in fluid flows around small hulls.

Moreover, numerical investigations have been performed considering the model in constant trim during its motion, contrast to the experimental case in which the particular towing system of the tank imposes a motion of the model in variable trim. Nevertheless, the authors is intended to limit the comparison on a qualitative reproduction of waves patterns and drag coefficients among experimental and numerical models; such comparison results satisfactory, as shown in Fig. 11, 12 and 13.

More critical dynamic condition occurs around maximum Froude number value; in order to get useful indications for hull form improvement, the attention is concentrated on the solution related to 0.51F_n (equivalent to speed of advance equal to 1.6 m/s for the model); which will characterize the following numerical simulations.

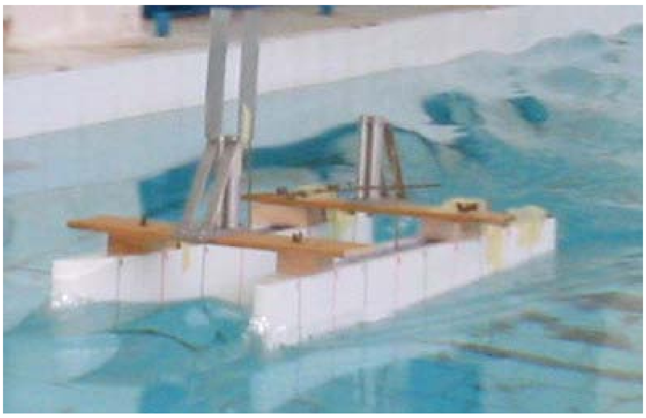

FIG. 12 EXPERIMENTAL TEST

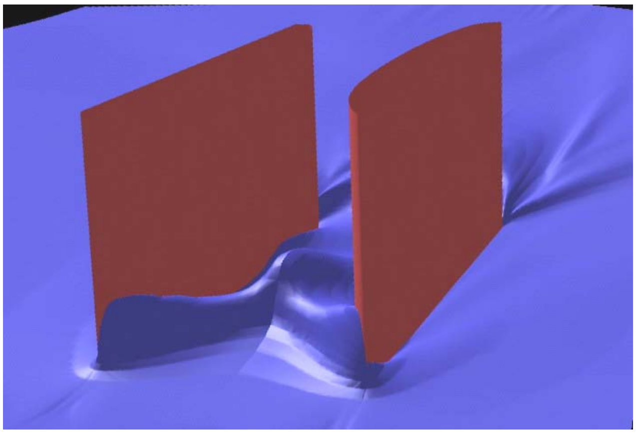

FIG. 13 PRELIMINARY SIMULATION

Hull shape optimization

The objective of an optimisation technique is to minimise a function depending on target values assigned to the main parameters. The optimiser, according to objective and constraint, attempts to modify the position of the control points. In such way a new set of control points is generated, and a new value of the optimising function can be established. The procedure stops when convergence is achieved (for a given tolerance) or some maximum number of iterations has been executed. At each main iteration, the algorithm writes the IGES file of the hull (in terms of NURBS curves and surfaces); and it is then possible to check the surface evolution during the optimisation process by means of any CAD software, (Mancuso, 2006; Augusto and Kawano, 1998; Calkins, Schachter and Olivera, 2001; Cappello and Mancuso, 2001; Cappello and Mancuso, 2004; Dejhalla, Mrsa and Vulkovic, 2001; Mancuso and Mancuso, 2004).

The optimization of shape proposed in this paper, consists in the research of catamaran geometry offering a motion total resistance lower than one of the preliminary numerical models, on the basis of a few main parameters like length of water line and displacement. During this process, the control points, that determine the form of the sections lines of the hull, are moved in order to increase the fillet on the inferior edge of the internal surface of preliminary hull (Fig. 14) with the aim of finding a solution that increases the hydrodynamic efficiency; and the displacement is maintained constant opportunely modifying the draft *T*. The shift of control points is bound to the respect of two conditions: the first maintains unchanged the design water line of the experimental model, fixing the position of control points of the design water line (Fig. 14); instead the second prevents that the control points assume positions determining the inversion of the curvature of the respective waterlines (Fig. 15).

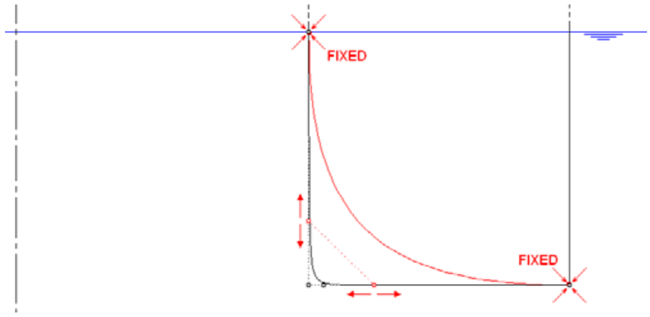

FIG. 14 CONTROL POINTS OF SECTION LINE

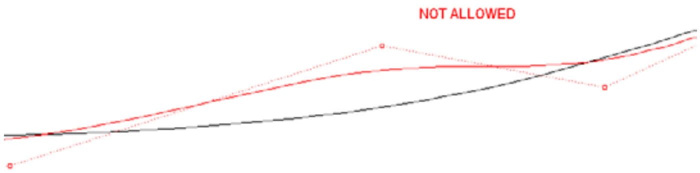

FIG. 15 CONTROL POINTS OF WATERLINE

At each step, hydrodynamic characteristics of the model are carefully analyzed and, on the base both of results and of the Authors experience, a new set of control points is fixed, producing a new model to examine in the subsequent step. This technique of shape optimization is not automatic, but it is very slim because it allows to limit the investigation only to the interesting solutions, since it is always under the control of the operator.

The whole procedure of domain building and meshing, CFD code configuration and execution of the numerical simulations has been planned in parametric way, with the purpose to automatically repeat it at every change whether of hull geometry or of tunnel width, during the optimization. Flow-chart in Fig. 16 reassumes the principal steps of this procedure. Replay files, shown in this scheme, are basically two programs written in the Tcl/Tk language containing all the necessary commands to run the complex meshing operations in batch mode; equally, the journal file contains the sequence of CFD code commands automating the numerical solutions. Numerous different versions of hulls were submitted to numerical verification by CFD code, following the same blocking strategy adopted in the preliminary phase, obtaining therefore grids with same number of nodes and same grading of preliminary model.

Finally the hull shape being able to offer the best hydrodynamic efficiency has been numerically investigated by varying the separation distance between hulls, according to Table 1.

Fig. 17 and 18 show the comparison of the optimized geometry with the preliminary one, in the hypothesis of beam equal to 0.400 m (case B). The reduction of the total resistance, achieved by the optimized hull, is about 20% in the whole range of simulated speed.

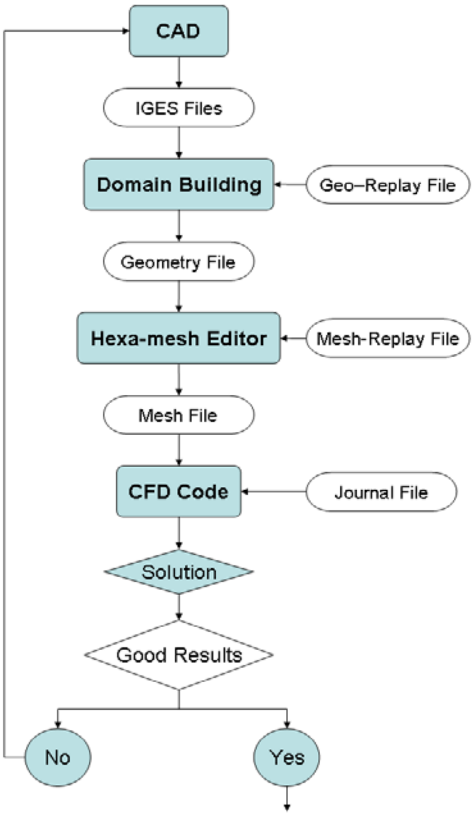

FIG. 16 OPTIMIZATION FLOW-CHART

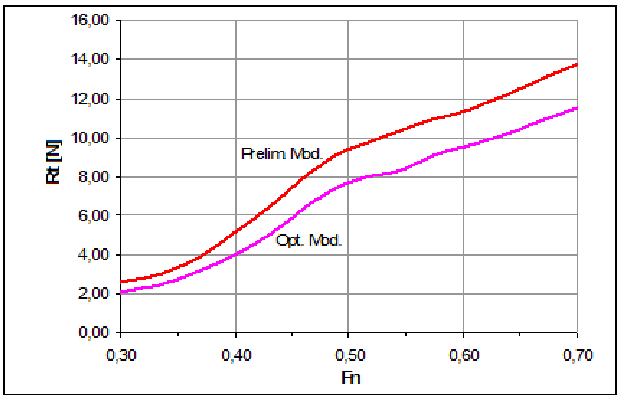

FIG. 17 TOTAL RESISTANCE CURVES - CASE B (B=0.400 M)

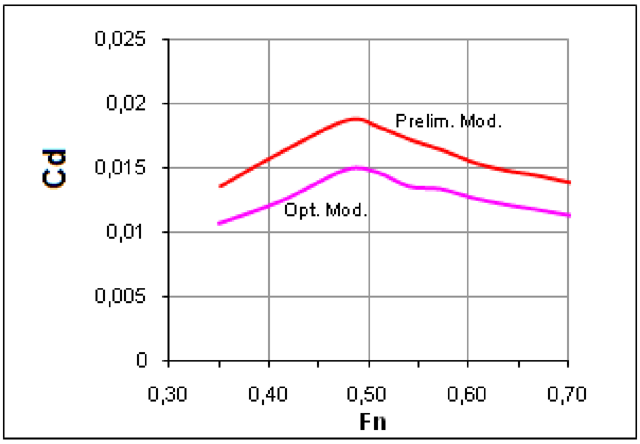

FIG. 18 DRAG COEFFICIENT - CASE B (B=0.400 M)

The comparison among the curves of resistance (Fig. 19), in the three considered conditions according to Table 1, shows that the configuration offering a greater hydrodynamic efficiency is characterized by beam equal to 0.440 m (case C), for lower Froude numbers than 0.67, while the configuration A, with beam equal to 0.340 m, has greater efficacy for higher Froude numbers.

Moreover, another important result is deduced by the Fig. 20 and, in particular, Fig. 21 that shows the curves of speed component v_x in the same direction of motion, respectively along the domain and inside the canal, obtained by numerical solution with an advance speed equal to 1.6 m/s (F_n = 0.51) in the case A. These curves prove the possibility to gain a partial recovery of the energy surrendered to the mass of water hypothesizing to install the propeller inside the tunnel.

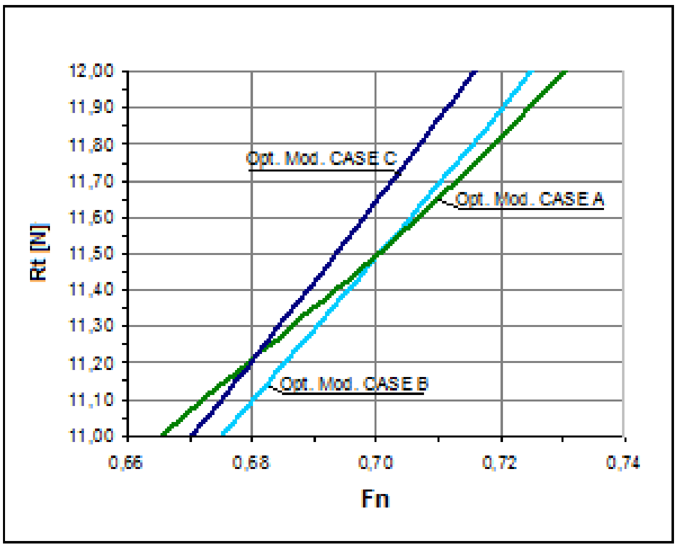

FIG. 19 TOTAL RESISTANCE CURVES COMPARISON

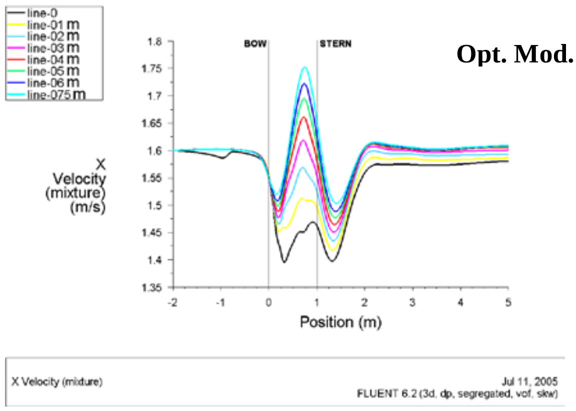

FIG. 20 CURVES OF SPEED COMPONENT V_X ALONG DOMAIN, CASE A (SPEED 1.6 M/S)

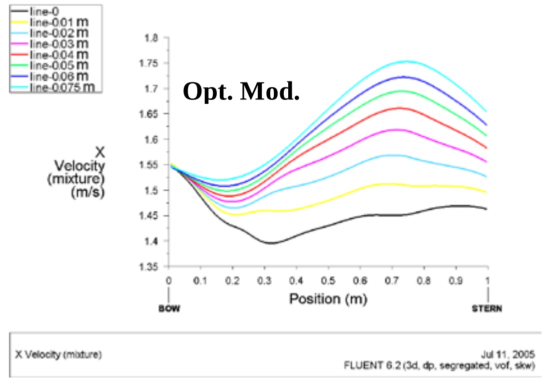

FIG. 21 CURVES OF SPEED COMPONENT V_X ALONG TUNNEL, CASE A (SPEED 1.6 M/S)

In fact, in correspondence of the section where water reaches the maximum speed, the kinetic energy of water can be converted into thrust energy, increasing the catamaran efficiency. However, the recovery of the kinetic energy has not been developed in this paper, therefore it requires further researches.

Fig. 22 shows the dynamic pressure on free surface: it puts in evidence the distribution of divergent and transversal waves. Fig. 23 shows the flow lines produced by the hulls of the optimized catamaran, in the configuration A, for advance speed equal to 1.6 m/s.

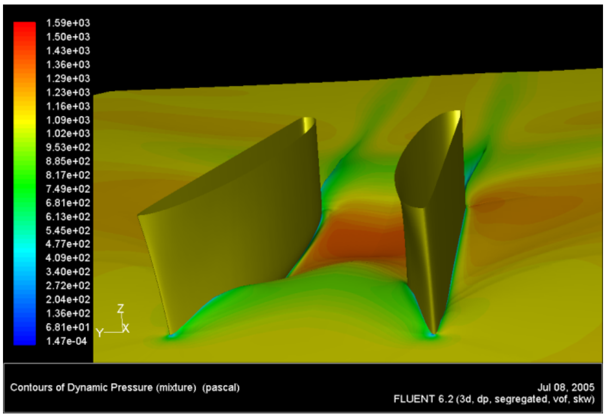

FIG. 22 CONTOURS OF DYNAMIC PRESSURE ON FREE SURFACE

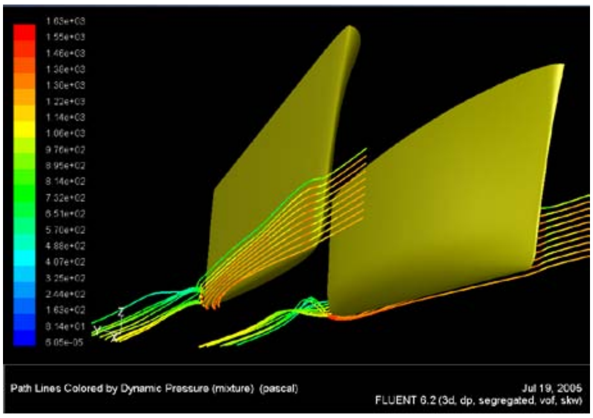

FIG. 23 FLOW LINES

Conclusion

The fluid-dynamic research presented in this paper has allowed the geometric definition of the proposed innovative catamaran, which maximizes the hydrodynamic efficiency of hulls. During the motion, the particular form of canal allows increasing the speed of fluid mass in the convergent line; and the kinetic energy of water can be exploited to advantage the catamaran efficiency. To verify advantageous effects of water acceleration inside the tunnel, a series of numerical simulations have been conducted using Fluent CFD code.

The experimental investigations have specifically concerned the wave patterns produced by hulls, for different values of the ratio between maximum width B_M and length water line L_{WL}. Some values of this ratio (0.340, 0.400 and 0.440) have been selected as configurations for CFD investigations. The preliminary numerical model of the catamaran was realized in order to geometrically result in what is similar to the tested experimental version, allowing calibrating the input data of CFD code.

Because the hull was destined to high-speed use, with Froude numbers superior than 0.50, the attention was put on the wave resistance due to the energy dissipated in wave making. Therefore great attention has been reserved both to modelling of free surface and to numerical evaluation of inertial and gravity forces governing the wave making, with the purpose to improve the calculation accuracy of the motion resistance by CFD analysis.

The three-dimensional optimization of hulls has allowed a reduction of total resistance equal to about 20% in comparison to the one of the preliminary geometry. The comparison of drag coefficients shows that the configuration offering a greater hydrodynamic efficiency is characterized by a ratio between beam and water line length equal to 0.440, for lower Froude number than 0.67, while, the configuration with ratio equal to 0.340 has greater effectiveness for higher values of Froude number. The results highlight significant differences between numerical and experimental results. It depends on the fact that the size of the tank is very small and some physical phenomena, as trim and sinkage, are neglected.

Curves of speed component v_x, (in the same direction of motion) inside the canal and along domain, in the case A, show the possibility to effect a partial recovery of the provided energy to the mass of water. In fact, hypothesizing the propeller installed inside the tunnel in correspondence of the section where water reaches the maximum speed, the kinetic energy of water can be converted into energy of thrust, increasing the catamaran efficiency. This problem is not treated in this paper and may be object of further research.

ACKNOWLEDGMENT

Authors thank "Istituto Tecnico Nautico G. Trabia", Palermo, Italy and Prof. G. Cambiano for the towing tank tests. The work was developed by grant of Italian Ministry of University and Research.

REFERENCES

Augusto O.B., Kawano A., 1998 "A mixed continuous and discrete nonlinear constrained algorithm for optimizing ship hull structural design", Ocean Engineering 25, 793–811.

Bertram V., 2002 "Practical ship hydrodynamics", Butterworth-Heinemann, Oxford.

Bruzzone D., Ferrando M., Gualeni P., 1999 "Numerical and Experimental Investigation Into Resistance Characteristics of Symmetrical and Unsymmetrical Catamaran Hull Forms", Proc. 5th Inter. Symposium on High Speed Marine Vehicles, HSMV99, Capri, Italy.

Calkins D.E., Schachter R.D., Olivera L.T., 2001 "An automated computational method for planning hull form definition in concept design", Ocean Engineering 28 297–327.

Cappello F., Mancuso A., 2001 "Lay-up optimisation for the hull of a racing sailing yacht", ADES, International Journal Advances in Engineering Software 32 (1) 133–139.

Cappello F., Mancuso A., 2004 "Curve and surface fitting via optimisation technique", CAD'04 International CAD Conference and Exhibition May 24–28, Pattaya Beach, Thailand.

Chapman R.B., 1972 "Hydrodynamic drag of semi submerged ships", ASME Journal of Basic Engineering,.

Couser P.R., Molland A.F., Armstrog N.A., Utama, I.K.A.P., 1997 "Calm Water Powering Prediction for High-Speed Catamarans", Fast '97, Sydney, Australia.

Damiano C., Lazzara S., Mancuso A., VirzìMariotti G., 2003 "Study on the Efficiency of an Innovative Hull", Nav 2003, International Conference on ship and shipping research, Palermo, June 24-27.

Damiano C., Stroligo M., Virzi'Mariotti G., Zotti I., 2009, Theoretical and Experimental Comparison among the Resistance Components of a Fast Catamaran in Different Configurations; NAV 2009, 16th International Conference of Ship and Shipping Research, Vol I, ISBN 978 – 88- 904394-0-7, Messina (Italy), 25-27 novembre 2009,pp 31-39.

Dejhalla R., Mrsa Z., Vulkovic S., 2001 "Application of genetic algorithm for ship hull form optimisation II", Shipbuild Program 48 117–133.

Kaklis P., Papanikolaou A., 1992 "The Wave Resistance of a Catamaran with Non-Symmetric Thin Demihulls", Proc 1st Nat. Congress on Computational Mechanics, Athens.

Larsson L., Baba E., 1996 "Ship resistance and flow computations", Advances in marine Hydrodynamics, M. Ohkusu (ed.), Comp. Mech. Publ.

Mancuso A., Mancuso U., 2004 "Ottimizzazione di Forma di Carene di Barche a Vela Mediante Analisi CFD, Il Progettista Industriale, 70–77.

Mancuso A., 2006 "Parametric design of sailing hull shapes", Ocean Engineering 33 234-246.

Michell J.H., 1898 "The wave resistance of a ship, Philosophical Magazine", Series 5, Vol. 45.

Papanikolaou A., Kaklis P., Spanos D., 1996 "On the wave Resistance of Twin-Hull Floating Bodies with Non-Symmetric Demi-Hulls", 11th International Workshop on Water Waves and Floating Bodies, Hamburg.

Schneekluth, H.,Bertram,V., 2002. "Ship design for efficiency and economy", Butterworth-Heinemann, Oxford.

Tuck E. O., Scullen C., Lazauskas L., 2002 "Wave Patterns and Minimum Wave Resistance For High-Speed Vessels", 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan.

Utama I.K.A.P., Jamaluddin A., AryawanW.D.; 2012, Experimental investigation into the drag interference of Symmetrical and Asymmetrical Staggered and Unstaggered Catamarans; The Journal of Ocean Technology, Peer-Reviewed Papers, Technology to the Wind, Vol. 7, No. 1, 2012, pp 47-58.

White, Frank M., 1991 "Viscous Fluid Flow", Second Edition, McGraw-Hill International Editions, Mechanical Engineering Series.

Wilcox D.C., 1998 "Turbulence Modelling for CFD", DCW Industries Inc., La Canada, California.

Zaraphonitis G., Spanos D., Papanikolaou A., 2001 "Numerical and Experimental Study on the Wave Resistance of Fast Displacement Asymmetric Catamarans", Proc. HIPER INT. Conference, Hamburg, May 2001.